

EnergyPlus Articles from the *Building Energy Simulation User News*

January 01, 2007

Through

December 31, 2007

**Simulation Research Group
MS: 90-3147
Lawrence Berkeley National Laboratory
University of California at Berkeley
Berkeley, CA 94720-0001**

January 2008

Copyright © 2007/08 The Regents of the University of California; pending approval by the U. S. Department of Energy. All rights reserved.

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Office of Building Technologies, Building Systems and Materials Division of the U.S. Department of Energy, under Contract No. DE-AC03-76SF00098. **Disclaimer:** This document was prepared as an account of work sponsored by the United States Government. While this document is believed to contain correct information, neither the United States Government nor any agency thereof, nor the Regents of the University of California, nor any of their employees, makes any warranty, express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process or service by its trade name, trademark, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof, or the Regents of the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof or of the Regents of the University of California

Table of Contents

Unitary Equipment.....	5
Water Heater Source Side Capabilities	5
Controls	5
Daylighting, Shading	6
Window Calculation Fatal Error	6
Connected Surface Warning.....	6
Zone Air – Heat Balance Equation.....	7
Domestic Hot Water Energy Use.....	7
Macros	8
Schedules	8
Residential Boiler.....	8
Fresh Air Question	9
Daylighting Obstructions	9
Schedules	10
Tall Building Simulation – Shortcuts.....	10
Heat Pump.....	10
Residential Boiler.....	11
Converting Doe-2 Binary Files Into Epw Format.....	11
Supply Fan And Return Fan Air Flow Rate.....	11
Cooling Tower.....	12
Glass Variables Defined	12
Otherside Coefficient.....	12
Façades	13
Zone Multiplier	14
Contiguous Zones.....	14
Heat Pump.....	14
Zone Multiplier – Uniform Zones	15
Floor And Roof Tilt – Part I.....	15
Floor And Roof Tilt – Part II.....	16
Illuminance Map.....	16
Chilled Beam System	16
Constant Deck And Supply Temperatures.....	17
Water-Side Economizer	17
Purchased Air And Ventilation	17
Gains From People, Lights.....	18
End-Use Category For Dx Coils	18
Light Shelf Surface Temperature.....	18
Meters	19
Reports -- Electric Energy Demand	19
Modifying Weather Files – Part I.....	20
Modifying Weather Files – Part II.....	20
Window Glass	20
Phase Change Material	20
Precipitation Data.....	21
Reporting Glare Index.....	21
Evaluating A Sun Shading Device.....	21
Dx Cooling Coil.....	22
Complicated Door-In-Wall	22
Sky Temperature	22
Autosizing Errors.....	24
Window Ac Or Unitary	24
Set Point Temperature	24
Multimode Humidity Control Issues.....	25
Fan Coil Unit With A Fresh Air Handling Unit.....	25
Airflow Network Model.....	25

Purchased And Outside Air	26
Generate A Dxf Drawing.....	26
Partitions And Volume.....	26
"Air" Partitions	27
Window Air Conditioner Nodes	27
Room Air Node Temperatures In Mundt Model	27
Direct Outside-Air System	27
Unit Conversion	28
Loads Not Met / Hours Not Met	29
Shading Devices	29
Window Reveals.....	30
Outside Reveal Shadowing.....	30
Zone With No Walls	31
Unentered Other Zone Surface.....	32
Corridor Modeling.....	32
Epw Files	32
Window With Switchable Glazing	33
Interior Blinds.....	33
Convert Doe-2 Fmt To Energyplus Epw.....	33
Airflow Object -- Design Volume Flow Rate	34
Model Creation And Autosizing	34
Autosizing Multiple Chillers	35
Autosizing And Parametrics	35
Wind Speed For Surface Orientation	35
System Volume For Chilled And Condenser Water	36
Loads Reporting.....	36
View Factors Error.....	36
Simulating An Extra-Light Structure.....	37
Wind Pressure Coefficients.....	37
Convection Coefficient	37
Hot/Chilled Water Reset Savings	38
Solar Output Variables	38
Sensible Heating And Cooling Rate For One Zone.....	39
Ground Temperatures And The Slab Program	39
Increasing Heat Transfer With Induced Turbulence	39
Inside Convection Coefficient	39
Thermal Comfort And Thermostatic	39
Supply Fan Variables	40
Zone Controls And Thermostats	40
Evapcooler Rdd Special Cooler	41
Floor Geometry.....	41
Heat Pump Data.....	41
Residential Heater Sizing And Cycling Time Step.....	41
Cooling Coil Error.....	42
Heater Element Exponent.....	42
Zero Air Loop Mass Flow Rate	43
Wet Bulb Temperature Inconsistency.....	43
Cfm/Ton Limits.....	43
Day-Of-The-Week Start Day	43
Warnings	44
Economics And Demand Cost Calculations	45
Latent Loads And Purchased Air.....	46
Purchased Air Heating And Cooling.....	46
Purchased Cooling Vs Fan Coil Cooling.....	46
How To Change The Default Output Units	46
Equipment Scheduling	47
Cooling Simulation	47
World Coordinate System.....	47
Warmest Function Modeling	48

U-Value Of Material Used	48
System Availability Manager.....	49
Naturally Ventilated Greenhouse.....	49
Free-Floating Simulation	49
Direct Air Model.....	49
Outside Air And Dx Coils	50
*.Rvi And *.Csv.....	50
Simulating Double Skin Façade.....	51
Window5 Data Vs Idf	52
Airflow Network Fan.....	53
Overlapping Shadows Error	54
Ventilation Objects	54
Curved Blinds	54
Exterior Fixed Louvers	55
Indoor Air Quality.....	55
Reporting Contaminants.....	56
Purchased Air And Infiltration	56
Doe-2 Translator For Energyplus	56
Solar Water Heating System.....	56
Shgc And Solar Transmittance.....	57
Chiller Operation.....	57
Dx Cooling Problem	57
Run Time Fraction	58
Surface Output Variables	58
Zone Sizing Output File	59
Hot Water Radiators	59
Index	61

UNITARY EQUIPMENT

Is there any way to convert a compact HVAC unitary system to a VAV system? I would like to simplify my input process by using this compact object, but I want to specify a VAV.

Answer

To model a unitary VAV system, it would be easiest to start with COMPACT HVAC:*:VAV objects and skip the plant loops. Run this (EnergyPlus will fail, but that's OK) and take the resulting expidf file and make the following changes:

1. In the main branch of the air loop, put in a DXSystem:AirLoop instead of the chilled water coil.
2. Delete the chilled water cooling coil object and its branch and controller, and delete the controller from the controller list.
3. Add a DXSystem:AirLoop and COIL:DX:MultiSpeed:CoolingEmpirical and related objects.

That should do it. Use example file 5ZoneAutoDXVAV.idf as a guide.

WATER HEATER SOURCE SIDE CAPABILITIES

I am trying to model the supply side of a heat recovery loop with a pump (as the supply inlet branch), a water heater (source side), and an exit pipe. Although the branches pass integrity testing, I receive a node connection error at the water heater inlet node with the statement that the inlet node did not find an appropriate matching "outlet" node. The same was true for the supply exit pipe. In the supply side of the loop, I have 15 m/s in-flow and 0 m/s out-flow. Does the source side of a water heater:mixed require a bypass?

Answer

No, a bypass is not required. You say the "branches" (plural) pass integrity testing. If you have the pump, the water heater and the pipe on separate branches, do you have a splitter connecting the inlet branch to the water heater branch? And a mixer connecting the water heater branch to the exit pipe branch? If not, this is most likely the source of the errors. An alternative approach is to place the pump and the water heater on a single branch (see example file TRHConstFlowChillerOneBranch.idf for a similar case with a chiller and pump).

Reply

Thanks! Creating a single branch worked.

CONTROLS

If I have a warm air system (fan, heater battery, supply ductwork) operating in winter to warm a factory to 20C during the day, but I want to avoid the temperature dropping below 12C overnight, can I control the fan and heater so that it is normally off overnight but comes on when room temperature drops below 12C?

Answer

Take a look at System Availability Manager:Night Cycle. The Input/Output Reference describes this manager pretty well. It will do what you want to do.

DAYLIGHTING, SHADING

I am simulating the daylight in three rooms of a three story house with almost-west facing windows. My model includes shading from a nearby building. I am comparing the average values of zone light maps for two conditions:

1. the existing shade building,
2. the existing shade building with a slightly higher roof.

Generally I have the expected result; that is, there is less light in my building when the shading building is taller. However, late in the year the first floor is indicating more light in the room for some hours when the shade building is taller. I don't see any input parameters for the shade building color. Is there a ground reflectance associated with the shade building that would cause more light to enter the first floor window when the shade building is taller?

Answer

That does seem non-intuitive. It is possible to add reflections from shading surfaces, but the default is no reflections (see BUILDING and SHADING SURFACE REFLECTANCE).

WINDOW CALCULATION FATAL ERROR

I'm trying to simulate a simple 5-zone single level Compact VAV block with windows and shades on all sides; however, the model failed to run stating:

```
*** Fatal ** Singular matrix in LUdecomposition, window calculations"
```

What am I doing wrong?

Answer

Remove the shades from the windows first to make sure it is not a problem with the windows. If the same error occurs you have a problem with the window construction. If the simulation completes successfully, add in a single shade and try again. Check the example file WindowTests.idf for examples of proper window and window/shade constructions.

CONNECTED SURFACE WARNING

EnergyPlus issues a warning if two connected internal surfaces do not have the same area; and I do understand that this should not be the case for most of the surfaces in a building. Can anyone tell me what actually happens within EnergyPlus in this case? Does it take the average surface area for the heat transfer calculation? Also, what happens if one of the surfaces has a different tilt than the other one?

Answer

EnergyPlus links the outside surface temperature for interzone surfaces. For example, assume Surface A and Surface B are linked. When Surface A is simulated, its outside surface temperature is set to be the inside surface temperature of Surface B. When Surface B is simulated, its outside temperature is the inside surface temperature of Surface A. Because the link is made by temperature, and not by heat flux, mismatched areas can result in violating conservation of energy. Mismatched tilt or azimuth will not cause an error if the areas match. However, error messages are generated because it may indicate that the wrong pair of surfaces has been linked to each other.

ZONE AIR – HEAT BALANCE EQUATION

I am trying to understand how EnergyPlus deals with the air heat balance equation. Even though it is described in detail I still don't know if the EnergyPlus heat balance equation takes into account the energy stored in the zone air.

Answer

Yes, the thermal mass of the zone air is included in the zone heat balance.

Question

And how do I plot the components of the air heat balance (assuming no heat transfer from inter-zone air mixing)?

Answer

You must compute the convective heat transfer from the zone surfaces by reporting Surface Inside Temperature, Surface Int Convection Coeff, and Zone Mean Air Temperature. The gain for each surface is $Area * ConvCoeff * (T_{surf} - T_{air})$.

DOMESTIC HOT WATER ENERGY USE

I'm trying to create a detailed model of annual domestic hot water energy use with EnergyPlus, similar to how it is done with DOE-2.

Answer

I'm assuming the example file you are talking about is DHWPurchHeating.idf which features the DOMESTIC HOT WATER object. To model hot water usage with this object, it is coupled as a component in the plant loop simulation. The plant loop supply-side equipment (usually a water heater or boiler) provides the heating and determines the loop hot water temperature. The DOMESTIC HOT WATER object sets the flow rate of water that is used and lost down the drain. The object also sets the return water temperature (presumably from the water mains), or if blank, the WATER MAINS TEMPERATURES object set the temperature.

If you are looking for something similar to the "DHW-" keywords under the PLANT-ASSIGNMENT command in DOE-2, I suggest using the WATER HEATER:MIXED object in "stand-alone" mode -- that means with no plant loop connections. See the example file WaterHeaterStandAlone.idf. In "stand-alone" mode, the water heater will accept a water use flow rate schedule that will allow you to set the DHW demand. See the documentation for more on this object. With the WATER HEATER:MIXED object you should find capabilities equivalent to DOE-2 and beyond.

For those EnergyPlus users who recall the energy balance bug that was identified with WATER HEATER:MIXED, please note that the object does work correctly when in stand-alone mode. The energy balance bug only occurs when the water heater is coupled to a plant loop. The WATER HEATER:MIXED bug was fixed in EnergyPlus 1.4; that means water heaters will function correctly for both stand-alone and plant-coupled applications.

MACROS

I am trying to create a macro that would run a number of simulations in EnergyPlus sequentially and save the output files, using unique file names, with each run. Ideally, I want the macro to:

- Change a certain number of parameters within an input file
- Simulate the input file within EnergyPlus
- Save the resulting output files, with specific names
- Begin the process anew

Is this do-able within EnergyPlus?

Answer

In the files area: http://tech.groups.yahoo.com/group/EnergyPlus_Support/files/Examples/ there is an example called Parametric-5ZoneWarmest.zip which will illustrate a simple way to do this using the macro capability and EP-Launch (which will take care of the separate folders for outputs).

SCHEDULES

I have a question about Schedules used with annual energy calculations, coupled with Design-Day calculations. For instance, if I have 100 zones and they all have the same annual schedule for heat gain, then I am able to define a single schedule and reference this schedule for all zones. If, however, I want to use different schedule data for the design summer or winter day for a room, this design-day data must be defined as a design-day schedule and referenced as the SummerDesignDay or WinterDesignDay in a WEEKSCHEDULE and then in a year SCHEDULE. If I have slightly different design-day schedules for each room, but the same annual schedules, must I then define different annual and weekly schedules for each room, in order to get the different design-day schedules defined? Or is there some other means of tagging rooms with just design-day schedule data and using common annual schedules for the annual calculation.

Answer

Your understanding is correct. You need a different annual/week schedule for each unique combination of design-day and regular-day schedules. You may find it easier to generate this using the SCHEDULE:COMPACT approach rather than DAYSCHEDULE/WEEKSCHEDULE/SCHEDULE.

RESIDENTIAL BOILER

I'm trying to model a simple residential house with three zones -- living, attic and garage. It's an older home with a central, coal-fired boiler used for heating. My problem is that I don't know how to 'connect' the boiler with convective heat exchangers in the three zones.

Answer

In the ExampleFiles folder, see WindACAuto.idf - Window air conditioners with hot water convectors served by purchased hot water (substitute a boiler for the purchased hot water). Run the example file and view the svg output file to see the types of components used to connect the hot water loop. Review the input file to see the input objects used to describe those components.
See also 5ZoneAirCooled.idf - Example with a boiler.

FRESH AIR QUESTION

I have a fan coil system with fresh air; chillers provide chilled water and the outside air flow rate (per person) is set in the zone sizing. Does the cooling capacity and energy consumption of the chiller include the capacity and energy that the fresh air needs?

Answer

The sizing of the fan coil's cooling coil chilled water flow rate takes into account the ventilation air. The chiller size will have built in the capacity necessary to meet the cooling load including fresh air. During the simulation the fan coil will act to meet its total load: zone cooling and outside air. This load will be passed to the chiller through the chilled water loop so the chiller will also meet the total load.

DAYLIGHTING OBSTRUCTIONS

Can EnergyPlus be used to model large internal obstructions to solar and daylighting penetration within a zone using surfaces or other objects? Clearly this would require the full exterior and interior solar distribution option, but I just wondered whether the presence of any internal obstruction surfaces would be accounted for in the calculation of daylight factors and solar distribution about the space. Are Daylighting:Detailed and Daylighting:DELight the same in this respect?

Answers

(From Mike Witte) -- I see that the daylighting section of the Input/Output Reference is inconsistent on these subjects. (On the list to fix.) Referring to p. 290 of the Input/Output Reference (pdf p. 327), internal obstructions accounted for in daylighting. This is not mentioned in the list of differences for DELight, so I assume that DELight also accounts for internal obstructions. I cannot say that I have verified this for either of the daylighting models. For solar gains, interior obstructions are not accounted for. The Full Interior solar distribution options require a "convex" zone. From p. 12 (pdf p. 49) "A formal definition of convex is that any straight line passing through the zone intercepts at most two surfaces."

(From Rob Hitchcock DELight expert) -- Regarding daylight (visible light) interreflections within a zone, the Detailed method uses a split-flux method, whereas DELight uses a modified radiosity method. The split-flux method divides transmitted daylight into "downward-going" and "upward-going" fluxes and evenly distributes a portion of these fluxes as reflected light based on an area-weighted average surface reflectance for the zone. See the Engineering Documentation page 116 (pdf page 148) for more detail. This method does not accurately account for interior obstructions in the interreflection calculation. The radiosity method more accurately accounts for interreflection between interior surfaces by discretizing each surface into nodal patch areas and accounting for the reflective exchange of visible light between the patch areas.

Regarding direct daylight distribution (i.e., exterior light directly reaching reference points and interior surfaces through apertures), the Detailed method does account for the obstruction, by the interior face of zone surfaces, of direct daylight on reference points (but not on interior surfaces), while the DELight method currently does not, assuming zones are convex, as Mike describes above for solar gains.

You might want to take a look at this paper

[DELIGHT2 DAYLIGHTING ANALYSIS IN ENERGYPLUS: INTEGRATION AND PRELIMINARY USER RESULTS](#) by William L. Carroll and [Robert J. Hitchcock](#), Proc. 9th International IBPSA Conference, Building Simulation 2005, August 15-18, 2005 in Montreal, Canada.

SCHEDULES

I have a question about Schedules used with annual energy calculations, coupled with Design-Day calculations. For instance, if I have 100 zones and they all have the same annual schedule for heat gain, then I am able to define a single schedule and reference this schedule for all zones. If, however, I want to use different schedule data for the design summer or winter day for room, this design-day data must be defined as a design-day schedule and referenced as the SummerDesignDay or WinterDesignDay in a WEEKSCHEDULE and then in a year SCHEDULE. If I have slightly different design-day schedules for each room, but the same annual schedules, must I then define different annual and weekly schedules for each room, in order to get the different design-day schedules defined? Or is there some other means of tagging rooms with just design-day schedule data and using common annual schedules for the annual calculation.

Answer

Your understanding is correct. You need a different annual/week schedule for each unique combination of design-day and regular-day schedules. You may find it easier to generate this using the SCHEDULE:COMPACT approach rather than DAYSCHEDULE/WEEKSCHEDULE/SCHEDULE. SCHEDULE:COMPACT is described in the EnergyPlus Input/Output Reference. There was also an explanatory article on p. 9 of the 2004 "Ask an EnergyPlus Expert" compilation, which can be found at this link: http://simulationresearch.lbl.gov/dirpubs/un_articleEP04.pdf.

TALL BUILDING SIMULATION – SHORTCUTS

I need to simulate an 80-floor skyscraper. I tried using a multiplier, but there is a differential load between different high floors. For example, there will be about 0.6C between 20m heights. And for so many floors, the gap cannot be ignored. It's a huge job if I have to simulate each floor, especially with an HVAC system. Also, run time is a problem. Are there any shortcuts I can apply to this problem?

Answer

From Peter Ellis, NREL, EnergyPlus Development Team

Not too long ago I had to deal with exactly the same problem in order to model the Freedom Tower at the World Trade Center (the previous twisty design, not the current one). The results of that exercise ended up going into a paper called "Simulating Tall Buildings Using EnergyPlus" that I presented at SimBuild 2005. The paper is available in the Conference Proceedings, or can be directly accessed at this link: http://www.ibpsa.org/proceedings/BS2005/BS05_0279_286.pdf

The surprising conclusion (to me anyway) was that despite height dependent variation of air temperature, wind speed, and shading from surrounding buildings, you get less than 1% error by simulating a single floor with a multiplier, e.g., 80, as long as you choose one near the mid-height of the building, i.e., floor number 40, when comparing whole-building annual energy results to a full simulation of all 80 floors explicitly. In the time since the paper was written, we have changed EnergyPlus so that, by default, the local air temperature and wind speed at a surface or a zone are now calculated based on its height above ground. We have also made it easier for the user to control the height dependent variation of atmospheric properties in EnergyPlus, even allowing height dependent variation to be turned off, if desired. See the WEATHER STATION object and SITE ATMOSPHERIC VARIATION object in the Input/Output Reference and Engineering Reference for more details.

HEAT PUMP

Can you tell me which field controls the heat pump's cooling/heating operation? Or are the modes simply shifted by the difference between the indoor air temperature and the setpoint? In my simulation of one internal zone and four external zones, I have one heat pump per zone. The heat pump in the internal zone will (hopefully) cool the space (which has large heat gains) in the winter while the other heat pumps in the outside zones are delivering heat.

Answer

The zone thermostat is the control. The heat pump will try to provide cooling when the thermostat calls for cooling, likewise for heating. One can schedule individual coils, so it is possible to disable cooling or heating with a schedule if desired.

RESIDENTIAL BOILER

I'm trying to model simple residential house with three zones -- living, attic and garage. It's an older home with a central, coal-fired boiler used for heating. My problem is that I don't know how to "connect" the boiler with convective heat exchangers in the three zones.

Answer

In the ExampleFiles folder, see WindACAuto.idf - Window air conditioners with hot water convectors served by purchased hot water (substitute a boiler for the purchased hot water). Run the example file and view the svg output file to see the types of components used to connect the hot water loop. Review the input file to see the input objects used to describe those components.

See also 5ZoneAirCooled.idf - Example with a boiler.

CONVERTING DOE-2 BINARY FILES INTO EPW FORMAT

I have some DOE-2 weather files in binary format (no text original inputs) that I'd like to convert to EPW format. Has anyone discovered a (hopefully) painless way to do this conversion?

Answer

Using the DOE2 WTH tool, convert to text format. The EnergyPlus Weather Converter reads that format and can convert into EPW. You will need to use the DOE2 bin to fmt converter first. All the utilities you need are in our "Yahoo Groups" file area; the folder name is DOE-2_Weather_Converter.

When you have the file in fmt format, the EnergyPlus WeatherConverter can convert to EPW format. You might also download the latest EPlusWth.dll; again, this is in our "Yahoo Groups" area in a folder Updates/Windows/.

Join the EnergyPlus Support group at http://tech.groups.yahoo.com/group/EnergyPlus_Support/

Caveat: Be aware that DOE-2 binary weather files save temperatures in integer F and solar radiation in integer BTU/ft², so any such file converted to EPW will lose precision compared to the original source data. In other words, the resulting decimal on these EPW files is good only to 0.3C precision, not 0.1C.

SUPPLY FAN AND RETURN FAN AIR FLOW RATE

I ran into a problem when I modeled a supply constant volume fan and return fan in the same air system. The air system uses 100% outdoor air flow with heat recovery. The supply fan air flow rate is 0.62m³/s. When I did not add the return fan, the system node check showed the system air flow rate was 0.62m³/s. To represent the real situation, a return fan was added to the air system with the fan air flow rate of 0.56m³/s (90% of supply flow rate, so that room has 10% positive pressure). However, I found that the system supply air flow rate was also reduced to 0.56m³/s. When I changed the return fan airflow rate, the system supply air flow rate stayed the same as the return fan airflow rate.

In my air system, the component configuration sequence is "return fan-cooling coil-heating coil-supply fan." The heat recovery unit is also in the air system but not listed in the air system branch. I have no ideal what is wrong with the configuration. I cannot find an example that includes both return fan and heat recovery unit in the air system.

Answer

EnergyPlus currently requires that all flow rates in an air loop be balanced. To model a system like this, you must specify zone exhaust fans which remove 10% of the flow directly from the zones. Set the ZONE EXHAUST FAN delta pressure to zero so that it does not consume additional electric power.

Question

So, the zone exhaust fan is the constant air flow fan module, right? If both supply and return fans are variable air flow, then the zone exhaust fan cannot be used. Can I use fan:simple:variable volume as the relief fans with delta pressure as zero?

Answer

You are correct, ZONE EXHAUST FAN is a constant volume fan. The flow rate can be scheduled, but it cannot track the system supply volume in a variable volume system.

A relief air fan will not accomplish what you want. For a variable volume system, there is no way to reduce the central exhaust flow by a percentage of total system flow. You will need to let the return fan have the same flow rate at the supply (non-pressurized system). By adjusting the return fan power and the effectiveness of the heat recovery unit, you can make the simulation behave as if there was 10% less return air flow.

COOLING TOWER

I am trying to model a Cooling Tower -- basically a chimney connected to the lobby at ground level with a wind catcher on top. There is also evaporative cooling in the tower with nozzles that spray water in the air. This cooler air is supposed to drive the air flow through the lobby. How do I account for this cooling effect in EnergyPlus?

Answer

An appropriate modeling strategy for the cooling tower would be to use a direct evaporative air system and then decrement out the associated fan energy. If the air flow rates match reasonably well, it should work okay.

GLASS VARIABLES DEFINED

Is there a document that defines each report variable? I can't figure out what is meant by just looking at the name; for example:

"Blind/Glass System Beam-Beam Solar Transmittance"

I don't know exactly what "beam-beam" means. Also, is visible light being transmitted for "transmittance," or is it heat, total energy?

"Blind/Glass System Diffuse-Diffuse Solar Transmittance"

Again, I'm confused by "diffuse-diffuse." And does the window system include the frame, the dividers, the blinds??

Answer

Refer to the Input/Output Reference document for a complete description:

Blind/Glass System Beam-Beam Solar Transmittance []

The fraction of exterior beam solar radiation incident on an exterior window with a blind (excluding window frame, if present) that is transmitted through the blind/glass system as beam solar radiation. Depends on various factors, including type of glass; solar incidence angle; slat angle, width, separation, and thickness; and horizontal solar profile angle (for blinds with horizontal slats) or vertical solar profile angle (for blinds with vertical slats).

Blind/Glass System Diffuse-Diffuse Solar Transmittance []

The fraction of exterior diffuse solar radiation incident on an exterior window with a blind (excluding window frame, if present) that is transmitted through the blind/glass system as diffuse solar radiation. Depends on various factors, including type of glass and slat angle, width, separation, thickness and reflectance. For blinds with a fixed slat angle the transmittance value reported here will be constant.

OTHERSIDE COEFFICIENT

I need to simulate a room that is located among three other classrooms (the other rooms are above, on the left and on the right). In order to establish exterior conditions for the walls connected to the other rooms, I defined air temperatures for the other rooms. I know that I need to define other parameters, such as convective/radiative film coefficient, etc. Do I have to define all of them?

Answer

It all depends on what you are trying to answer in your simulation. If the surrounding spaces are similar to the classroom you want to model, and are conditioned to the same temperatures, then you may want to make the interior surfaces adiabatic. For example Surface:HeatTransfer, Name=Wall1,

OutsideFaceEnvironment=OtherZoneSurface, OutsideFace Environment Object=Wall1. Or you can model the surrounding spaces in their entirety as separate thermal zones if the conditions in those zones will be significantly different.

FAÇADES

I need to model two façades for a 10-story building. One façade area is almost 100% glass, while the other is composed of a frame and divider that is part glass and part sheet metal.

Answer

In EnergyPlus, all windows must be modeled as a subsurface (Surface:HeatTransfer:Sub) that is placed on a base surface (Surface:HeatTransfer). If you make the base surface material match the properties of the frame, then you can place windows on the base surface to cover almost all of the surface (however, there must always be a small amount of base surface area remaining). For the façade with dividers, you can either place multiple windows on the base surface, or place one large window and use WindowFrameAndDivider to add the dividers. Note that the frame area of WindowFrameAndDivider is added outside the Surface:HeatTransfer:Sub area.

ZONE MULTIPLIER

I simulated a 7-story building, with each floor having two zones. I used a zone multiplier to multiply my middle floor by 5 times. Do I need to divide every resulting value for the middle floor by 5 in order to obtain a value just for a typical middle floor?

And about infiltration. Should I input infiltration values only for a typical middle floor, assuming that EnergyPlus automatically multiplies the given infiltration value by 5 times, or do I need to input an infiltration value 5 times that of a typical middle floor?

Answer

Those are good questions. The zone multiplier is applied when loads are passed to the HVAC system and when energy use is metered. For the energy meters, you can see in the MTD file where the multipliers have been applied.

For example:

```
Meters for 5037,PER-1I LIGHTS 1:Lights Electric Consumption [J] * 4 (Zone Mult = 4, Zone List Mult = 1)
```

In general, zone-level objects, such as lights, people, infiltration should not be multiplied upon input and will not be multiplied upon output. Only HVAC output and meter output is multiplied. So, you do not need to multiply the infiltration values. And the window heat gain should be the un-multiplied value. But I would recommend changing the multiplier and confirming that the window heat gain does not change.

CONTIGUOUS ZONES

Is it possible to create two contiguous zones in such a way that there is no surface dividing them? I want to do this in order to visualize the difference between thermal comfort of the people who work next to the windows and those who work in the core of the building, away from sunlight.

Answer

Not really. You can define two zones without a dividing surface, but there will be no exchange between the zones where they meet. If you add an interzone window, you can pass solar gains from one zone to the next. And if you add MIXING or CROSS MIXING, you can create an air exchange between the zones. But there is no radiant exchange from one zone to the next. If you want to report thermal comfort for two locations, it would be better to model the two areas in the same zone and use two different PEOPLE objects. By using Angle Factor List(s), you can change the mix of surfaces by which each PEOPLE object is affected. Note that solar gains incident on the PEOPLE are not included in the thermal comfort models. Only the inside surface temperature of the window impacts the reported thermal comfort values.

HEAT PUMP

Which field controls the heating/cooling operation of the heat pump? And are the modes shifted by the difference between the indoor air temperature and the setpoint? In my simulation (one internal zone and four external zones) I assigned one heat pump per zone. I wanted the heat pump in the internal zone to cool the space (which has large heat gains) in the winter, while the others in the outside zones deliver heat.

Answer

Heat pump is controlled by the zone thermostat. The heat pump will try to provide cooling when the thermostat calls for cooling, likewise for heating. You can schedule individual coils, so it is possible to disable cooling or heating with a schedule if desired.

Question

If I may continue with this theme: If a center zone still demands cooling from the individual heat pump during a cold winter, how does the Heat Pump deliver the cooling air? The supply air in question has a temperature lower than the ambient temperature (-5C). Shouldn't the supply air in the evaporator always be cooler than the outdoor condenser in the cooling cycle? Or is the heat pump using imaginary cool air to meet the cooling load against possible ice-cold air coming into the zone?

Answer

In this case, the supply air temperature will be whatever the mixed air temperature happens to be. If the outside air fraction is high, then this system will deliver very cold air to the zone. If you anticipate that this might be too cold, then a preheat coil must be added to warm the cold outdoor air or to warm the mixed air.

ZONE MULTIPLIER – UNIFORM ZONES

If I have several zones that are going to have the same loads (i.e., similar sized zones with exactly the same number of people in each, on the same schedule, etc.) is there a way to assign multiple zones to one load definition object? I want to avoid having to come up with unique names for each load object.

Answer

Not exactly, you'll need separate People objects with unique names for each zone (try using the zone name as part of the name to get unique names).

If your zones are really uniform (same sizes, internal gains, orientation, etc.), then you could go an even simpler route, which is to use zone multipliers. This way, you define a single zone just one time and then give it a zone multiplier. The resulting loads will be amplified by the multiplier.

FLOOR AND ROOF TILT – PART I

How do I check that the tilt for floors and roofs is correctly defined?

Short Answer

Put in a Report, Surface, Details; and review the results in the eplusout.eio file. That will tell you the computed tilts and facing (azimuth) for all surfaces. Results are easily importable into a spreadsheet program.

Longer Answer

For Tilt of roofs and floors, you have to specify the coordinates in the order you have given in the "SurfaceGeometry" object. All of the EnergyPlus example files use "counter clockwise" to specify the coordinates. If you specify them "clockwise" instead, you will get a 180 degree tilt from what you want. Or another way to think of it is to "look" from the outside of the roof and specify the coordinates that way.

For instance, if you have a flat roof at 4.5 meters above horizontal, specifying (counter clockwise):

```
(0.0 , 15.24 , 4.5)
(0.0 , 0.0 , 4.5)
(5.24 , 0.0 , 4.5)
(15.24 , 15.24 , 4.5)
```

will give you a flat roof of the proper tilt.

whereas:

```
(0.0 , 15.24 , 4.5)
(15.24 , 15.24 , 4.5)
(5.24 , 0.0 , 4.5)
(0.0 , 0.0 , 4.5)
```

will give you a roof of improper tilt. Look up Tilt or SurfaceGeometry in the documentation for an extended explanation.

FLOOR AND ROOF TILT – PART II

The EnergyPlus example files show differing tilt angles for ceiling and similar for floor. What should the tilt angle be for a floor and a ceiling and how does this affect the calculations?

Answer

Floors are usually flat and tilted 180 degrees. Ceilings would usually be flat as well and tilted 0 degrees. Roofs, of course, can be gabled and, therefore, tilted at other angles. Since EnergyPlus uses coordinate input, it derives the "tilt" angles from those. Tilt angle will affect calculations since tilt angle will be the "outward facing angle" for the surface. Floors that are tilted 0 degrees can't view the walls that surround it, for example. Azimuth or facing angle is also derived from coordinates and depends (for floors and roofs) on which corner is selected.

ILLUMINANCE MAP

I'm trying to work with the illuminance map, on a model with white walls and ceiling. But the values are too high even when there isn't direct sunlight. For example, at the end of the day, in the back of a 5m room, there is a value higher than 800lx. What could be the problem? And does the program consider external reflectances? For example, the reflected light at a nearby building?

Answer

Check the coordinates for the illuminance map, noting that all daylighting coordinates are relative to the zone origin (even if WorldCoordinates is specified for geometry.)

EnergyPlus can model external reflectances, but this feature is off by default. See BUILDING "Solar Distribution," and Shading Surface Reflectance.

One other factor contributing to the higher than expected values you are seeing "at the back of the room" might be the split-flux method used to calculate the daylight contribution to interreflected light at reference points. This method "splits" the incoming daylight into "upward" and "downward" proportions and then evenly distributes the interreflected light based on area weighted average surface reflectances in these portions of the space. The result is that the interreflected contribution is the same at all reference points, regardless of location in the space. If this component is large relative to the direct contribution, as with highly reflective surfaces (white walls and ceilings) and no direct sunlight, this can skew the results.

CHILLED BEAM SYSTEM

I need to model the chilled beam system using EnergyPlus. Any hints?

Answer

Unfortunately, at this time there are no chilled beam examples. It has been suggested that chilled beam systems can be modeled using the low temperature radiant system (with chilled water in the slabs) and then manually increase the convection coefficient on the ceiling surfaces (using the CONVECTION COEFFICIENTS object). For an example of radiant slab cooling systems, see ExampleFiles\RadLoHydrHeatCoolAuto.idf.

Alternatively, chilled beams could be modeled as fan coils with zero fan power, since their output is predominantly convective.

CONSTANT DECK AND SUPPLY TEMPERATURES

I am trying to simulate a 100% outside air, constant-volume, constant-temperature ventilation system. I want to dehumidify the ventilation air by using a DX coil (I used a DXSystem:AirLoop object) and then a gas reheat coil (FURNACE:BLOWTHRU:HEATONLY). I want the air to come off the DX coil at a maximum and constant 55F (except during the winter when the OA to the system will be lower than that temperature). I want the air to come off the gas coil in the furnace at a constant 68F, year round, no matter what. I tried using setpoint controllers at the outlet nodes to both coils, but the furnace mandates that the thermostat location be in a zone; this causes my setpoint managers to be over-ridden by the zone needs.

Answer

While the DX coil cannot control itself directly, a gas heating coil can. For this application, delete the FURNACE object and put the gas heating coil directly on the branch and then specify the coil outlet node to be the Coil Temp Setpoint Node. Use SET POINT MANAGER:SCHEDULED to place the heating and cooling setpoints at the respective outlet nodes.

I have made a note to make sure that the Input/Output Reference Manual explains which coils can go directly on a branch, which need a controller, and which need to be contained inside a parent object.

WATER-SIDE ECONOMIZER

I am trying to use the water-side economizer in EnergyPlus by adding a Free Cooler Branch: HEAT EXCHANGER:HYDRONIC:FREE COOLING. I referred to the example file (Free cooling chiller.idf) but there are some things I don't fully understand. Here are my questions (and thank you in advance);

1. What is the difference between Chiller: Electric and Chiller: Const COP? In real applications, which one is more practical?
2. When I use the CHILLER:Electric object, I am not quite sure about some of the parameters. For example, say the chiller supply water is 6C and the COP is 3.2. If I increase chiller water temperature to 15C (used for radiant cooling), should I change the COP to about 4.3 in the model input? Also, should I change the coefficients inputs? And if I have to change the coefficient inputs, where do I get some reference data?

Answer

1. Const COP chiller is an ideal chiller. Use CHILLER:ELECTRIC or CHILLER:ELECTRIC:EIR
2. The CHILLER:ELECTRIC and CHILLER:ELECTRIC:EIR models automatically adjust for differing chilled water and condenser water temperatures. The design (or nominal) COP entered for the chiller must be consistent with the design chilled water and design entering condenser temperatures. You can report the chiller COP using REPORT VARIABLE "Chiller COP."

PURCHASED AIR AND VENTILATION

I'm modeling a restaurant that has three air handling units and some exhaust fans in the kitchen. I want to use the PURCHASED AIR object, and I have information on the building's Outdoor air, supply air, return air and the Kitchen's exhaust air. Where should I specify each one of those and which flow rate should I specify for the Ventilation object?

Answer

If you want the EnergyPlus simulation to always meet the load, then allow the PURCHASED AIR object to have unlimited capacity.

If you want to limit the PURCHASED AIR flow rate, then specify the supply air flow rate.

Assuming the systems are balanced such that the outdoor air flow is equal to or greater than the exhaust fan flow, then specify the outdoor air flow rate in the VENTILATION object(s).

If you are modeling three thermal zones to align with the three air handlers, then you may also want to add a MIXING object if there is a net flow of outdoor air from the dining area(s) into the kitchen.

All of this assumes that the air handlers will be constant volume with continuous fan during occupied hours.

GAINS FROM PEOPLE, LIGHTS

How can I find the exact magnitude of gains (Watts) due to people, due to lights, etc.

Answer

In general, you want to check out the .rdd file (created by EnergyPlus) for a complete list of report variables available for the model. Then repeat the run with additional requests for specific report variables in which you are interested. Requests for results can be for many different time scales and appear in different places. First, get the .rdd file and include this in your input file:

Report, Variable Dictionary;

Also, we would suggest adding the following report requests to your input file. These next two objects will produce a separate "Tabular" report file that may have all the information you need.

Report:Table:Style, HTML;

Report:Table:Predefined,

Annual Building Utility Performance Summary,

Input Verification and Results Summary,

Equipment Summary;

For time series results you could use the following
(you could convert from J to W using the time step):

Report Variable, *, Zone Lights Electric Power, Timestep;

Report Variable, *, Zone People Total Heat Gain, Timestep;

(you could convert from W/m² to W using the surface area).

Report Variable, *, Surface Ext Solar Beam Incident;

END-USE CATEGORY FOR DX COILS

For LEED M&V purposes I would like to assign a specific end-use category to the energy consumption associated with DX coils (compressors, condenser fans). However, I cannot find the option. Am I missing it somewhere?

Answer

Custom end-use categories are only supported for a limited set of components at this time. You can, however, create custom meters. The following object would collect all DX coil energy use (compressors and condenser fans) onto a single meter:

CUSTOM METER,

Custom DX Cooling Meter, !- Name of Meter

Electricity, !- Fuel Type

*, !- Key Name #1

DX Cooling Coil Electric Consumption; !- Report Variable or Meter Name #1

This will not add a new line to the ABUPS end-use table, but it can be reported just as any other meter:

Report Meter,

Custom DX Cooling Meter, !- Meter_Name

monthly; !- Reporting_Frequency

Connections to all meters (master, end-use, and custom) are shown in the mtd output file.

LIGHT SHELF SURFACE TEMPERATURE

I want to simulate the energy performance of a light shelf and I'm facing the following problem: the surface temperature of the internal portion of the shelf is very high. For example under the climate conditions of San Francisco, and when I'm using the provided example "DaylightingDeviceShelf.idf," the surface temperature of the shelf reaches 68C. This fact also influences the cooling (increased) and the heating (decreased) loads, due to the existence of a heat source into the space. Do you have any ideas on how to solve this problem?

Answer

The problem is that the input file "DaylightingDeviceShelf.idf" was designed to be the simplest example of how to use the DAYLIGHTING DEVICE:SHELF object. However, the input file does not necessarily represent a realistic building model. If you check the zone temperatures, they are also very high, peaking at 41C. The reason for all the hot temperatures (and cold ones in the winter) is that these zones are not conditioned; i.e., they do not have any heating or cooling systems. Nor do they have any mechanism for exchanging air with the outdoors -- no infiltration or ventilation. Therefore, because of the windows, you have a lot of solar gain in the summer with no way to remove the heat (although there is no insulation in the walls and the windows are single pane, so that helps a little). All of these things were left out for the sake of simplicity of the example.

The surface temperature of the inside shelf is naturally going to be even hotter than the zone because the direct beam of the sun strikes the surface during much of the day. (By the way, this probably does reflect the real behavior of light shelves.) I would expect that the inside shelf and the floor will be the hottest surfaces in the zone, which then give off their heat to the zone air by convection and to other surfaces by radiation. One thing that the light shelf model WILL NOT capture is the "fin effect" of an outside shelf. For example, if you have a light shelf constructed from a single slab of material, and it forms the inside part and the outside part of the same shelf, there will be some conduction losses that will cool down the inside shelf, assuming that it is hotter than the outside. So you might find that the simulated inside shelf is slightly hotter than reality. If that's a problem, you can decrease the Solar Absorptance of the finished surface material of the inside shelf; it won't absorb as much solar and won't heat up as much.

If you are interested in simulating the energy performance of a real light shelf, I recommend that you study the example file to learn how to use the DAYLIGHTING DEVICE:SHELF object. Then, create your own model of a realistic building or zone to which you can add the DAYLIGHTING DEVICE:SHELF. You should get good results that way.

METERS

In the meters report, is "Purchased Cooling:Facility" the same as the Electricity consumption of the HVAC system?

Answer

No. "purchased cooling:facility" is cooling energy obtained from some off-site source of cooling. It applies to object types PURCHASED AIR and PURCHASED:CHILLED WATER.

To understand the meaning of each meter and submeter, see the .mtd output file. The first half of this output file lists every metered report variable and the meters to which it is connected. The second half provide the inverse, listing all the report variables connected to each meter. Reviewing this output file should help you understand the meaning of the various meters.

REPORTS -- ELECTRIC ENERGY DEMAND

I have a general question. How is COOLING:ELECTRICITY {MAXIMUM}[W] in Report:Electric Energy Demand (ABUPS) related to Peak DES COOL LOAD (W) in the EnergyPlus zsz (or eio file)?

Answer

"Peak DES COOL LOAD (W)" in the zsz and eio output files is the peak sensible cooling requirement of the zone as computed for the design days in the input file.

"COOLING:ELECTRICITY {MAXIMUM}[W]" in Report :Electric Energy Demand (ABUPS), is the peak power consumption of all electric cooling equipment in the simulation over an entire weather file simulation. To see what specific components are part of COOLING:ELECTRICITY, see the mtd (meter details) output file.

MODIFYING WEATHER FILES – PART I

I am studying building envelope thermal loads, comparing simulation with on site data. The building in question has a weather station on top and I would like to run a simulation with this weather data for a period of one week only.

Answer

Look at the Auxiliary Programs guide. It is the document that discusses the weather converter and explains how to create a custom file in detail.

Here is how to make EnergyPlus run for just one week or one month:

Enter the appropriate dates in the Run Period object, making sure that your Run Control field says to run the weather simulation. You can run more than one run period (i.e., you could run several-week run periods in a single simulation). The weather file must be for one year. Of course the simulation can be for any period. If you only have 1 week of weather data, fill in the rest of the year with whatever you want -- you could use constant conditions or replicate the week you have 52 times.

MODIFYING WEATHER FILES – PART II

How can I modify a weather input file just for a particular period of simulation and for a particular variable? Data from weather files are ALMOST the same as those obtained from direct measurements, more precisely wind speed and direction. I would like to modify these two values in order to get better results. Other variables, such as relative humidity and solar radiation are fine; that's why I need to modify only a couple of the parameters.

Answer

I'd suggest running the weather file through the WeatherConverter (Weather Statistics and Conversions under EnergyPlus Programs from the Start button) to change it to a CSV (comma-separated values) file. Then you can open and modify it in Excel. This CSV includes headings for the hourly data. Save it back to CSV, then run it through the WeatherConverter again to get your revised EPW.

WINDOW GLASS

I am simulating a series of windows with different U-values (conductance, w/m^2-k). However, the input of WindowGlass requires Conductivity ($w/m-k$).

Answer

Conductance is Conductivity divided by Thickness, but this will give you the conductance of just the glass (and gas) layers only. I presume that the U-values you are referring to also include the inside and outside film (convection) coefficients as well. If they are from Standard 90.1 (or 90.2) they use the film coefficients for a specified set of temperature and wind-speed conditions. EnergyPlus will report this standard U-value in "Report, Surfaces, Details;" See Output Details and Examples document (p. 23, PDF see p. 29) for more information. You will need to use the film coefficients shown there, remove them from the U-value, then calculate the thermal conductivity that will result in the desired U-value.

PHASE CHANGE MATERIAL

Is there an example file for phase change materials?

Answer

In Version 2.0 (and above), refer to this: CondFD1ZonePurchAirAutoSizeWithPCM.idf.

PRECIPITATION DATA

I want to examine precipitation at a number of sites and hope to use the data in some epw files. However, I am puzzled by some of the numbers in the data column that correspond to precipitation. The documentation says the data has units mm over the hour in question. However, when looking at the data series for both Phoenix and Seattle the numbers are in the range 5 to 25 and there are no zeros; this suggests there is continual rain. If this range was in mm of precipitation it would amount to a huge rainfall over the year.

Answer

That field is not precipitation. Note from our description of the EPW fields:

Field: Precipitable Water

This is the value for Precipitable Water in mm. (This is not "rain" – rain is incorporated in the preceding two fields). It is not currently used in EnergyPlus calculations (primarily due to the unreliability of the reporting of this value). Missing value is 999.

We use precipitation data sets as long schedules. Examples of these are included with the DataSets\PrecipitationSchedulesUSA.idf. These were modeled from NOAA monthly average rainfall totals with rates and timing adjusted to match the integer codes you are seeing. See the TMY2 weather documentation. Here's the link for the TMY2 manual: <http://rredc.nrel.gov/solar/pubs/tmy2/PDFs/tmy2man.pdf>

REPORTING GLARE INDEX

I have been simulating the influence of dynamic shading devices on overall electrical consumption, using monthly data reporting. As might have been expected, when using the control "shading device on if high glare," the electrical consumption for lighting rose. But the reported DGI's are averaged, so absolute numbers are well below the index that triggered the control. Therefore, as a suggestion, would it be possible to include a reporting variable that indicates when daylight glare index was not met, as well as the maximum glare index?

Answer

You can report maximum glare index using REPORT:TABLE:MONTHLY. You can also use REPORT:TABLE:TIMEBINS to determine the number of hours the glare index was not met. Set the upper limit of the bin range to equal the glare index setpoint, and then the last column will report the number of hours greater than that value.

Assuming your shading devices can completely cover the window, there should not be any hours where the glare index was not met. If I understand the control algorithm correctly, it first checks for glare levels, then deploys shades as needed, then simulates for that timestep. So, the only times the glare index limit would not be met would be if the shades cannot fully close (in the case of blinds) or the shades are not dark enough or do not cover all windows.

EVALUATING A SUN SHADING DEVICE

I am doing research on sun shading devices and I am facing some problems. I have developed a preprocessor that builds a solar shading device with slats using the Surface:Shading:Attached model to define the slats as individual surfaces. I would like to compare the results of my sunshading with the results I get from the Material:Window:Blind object to simulate Venetian blinds. All the dimensions in both objects are similar, yet the inside temperatures resulting from the two models differ from 1C to 2C. I looked through the Engineering Reference but couldn't find out what causes this difference:

Answer

The modeling of surface:shading:attached is very simple compared to the model for Material:WindowBlind. Shading surfaces can both shade and reflect. There is no heat balance on shading surfaces. Shading surfaces are generally only useful on the exterior;

I assume that is where you are modeling this. External shading surfaces do not block the wind imposed on the exterior surface of the window, but exterior blinds do alter the exterior window convection coefficient. The only result I would expect to be comparable between these two approaches would be the transmitted solar through the window.

DX COOLING COIL

I am trying to simulate a rooftop VAV system with DX cooling. What is the best way to simulate a rooftop with four DX compressors, each one representing 25% of the total capacity?

Answer

It depends on how the stages are arranged and controlled. If the four stages are all in parallel (four sections of face-split coils), then the best you can do is to model this with two stages using the multimode DX coil. If the staging is through cylinder unloading, then it's more of a PLR curve issue than face-split stages.

COMPLICATED DOOR-IN-WALL

There is a door across the top and bottom of a wall. However, both top and bottom have different constructions and have been specified with four vertices each. I'm getting an error file.

Answer

If you have split your surface into two different constructions (and surfaces), then you will have to also split your door into two surfaces (with the same construction). Each surface of the split door should lie within the vertices of the corresponding surface to which it is attached.

SKY TEMPERATURE

How does Energy Plus compute sky temperature for long-wave radiation heat transfer with exterior surfaces?

Answer

It uses the Horizontal IR from sky as a basis (we calculate this on the weather files and design days)

```
! Calculate sky temperature, use IRHoriz if not missing
IF (IRHoriz == "missing") THEN
! Missing, use sky cover
OSky=OpaqueSkyCover
TDewK=MIN(DryBulb,DewPoint)+TKelvin
ESky= (.787 +.764*LOG((TDewK)/TKelvin))*(1. + .0224*OSky -
0.0035*(OSky**2) + .00028*(OSky**3))
SkyTemp=(DryBulb+TKelvin)*(ESky**.25)-TKelvin
ELSE ! Valid IR from Sky
SkyTemp=(IRHoriz/Sigma)**.25 -TKelvin
ENDIF
```

Sky IR is calculated as:

```
! Determine Sky Temp ==>
! Function of DryBulb, DewPoint, OpaqueSkyCover
! Calculate Sky IR
!HIR = ESKY * SIGMA * (TOUT**4)
!
!where
!
!HIR = horizontal IR intensity (W/m2)
!ESKY = sky emissivity
!SIGMA = Stefan-Boltzmann constant = 5.6697e-8 W/m2-K4
!TOUT = drybulb temperature (K)
!
!The sky emissivity is given by
!ESKY = [0.787 + 0.764*ln(TDEW/273)]*[1 + 0.0224*N -
0.0035*(N**2) + 0.00028*(N**3)]
!
!where
!TDEW = dewpoint temperature (K)
```

!N = opaque sky cover (tenths)
!
!Example: Clear sky (N=0), TOUT = 273+20=293K, TDEW = 273+10=283K:
!
!ESKY = 0.787 + 0.764*0.036 = 0.815
!HIR = 0.815*5.6697e-8*(293**4) = 340.6 W/m2

References:

- George N. Walton, "Thermal Analysis Research Program Reference Manual," NBSIR 83-2655, March 1983, p. 21.
- G. Clark and C. Allen, "The Estimation of Atmospheric Radiation for Clear and Cloudy Skies," Proc. 2nd National Passive Solar Conference (AS/ISES), 1978, pp. 675-678.

AUTOSIZING ERRORS

I used autosizing for systems, equipment and plant. The setpoint air temperature was 24C for cooling and 22C for heating, and I had a VAV system serving five zones. When I used autosizing for humid locations (like Portland or Atlanta) the simulated indoor operative temperature for perimeter zones was well within the 25-26C range; however, when I changed the weather file and design day file for dry climates like (Helena or El Paso), the indoor operative temperature in perimeter zones reached from 28-30C.

Answer

Operative temperature is a mix of zone air temperature and zone surface temperatures. If zone air temperatures are meeting setpoint, then you should look at the results for mean radiant temperature and the temperature results for the inside face of surfaces. If you have some large windows, then they could be getting hot when in direct sun and will cause operative temperature to be quite high.

If you are concerned about meeting operative temperatures rather than air temperatures, then you can use operative temperature-based thermostatic control. Systems usually autosize to be larger with the operative temperature control.

WINDOW AC OR UNITARY

I need to verify EnergyPlus simulation results by taking actual measurements with an anemometer and temperature sensors in a room with an evaporative air conditioner inside the room and an air cooled condenser outside the room. There are examples for both Window AC and Unitary equipment class; which one is more appropriate to this split type of air conditioner?

Answer

Both the Window AC and the Unitary System objects use the DX cooling coil object Coil:DX:CoolingBypassFactorEmpirical. This cooling coil object is probably what you will need to use -- cooling only, single stage. It would probably be easiest for you to use the Window AC example. In EnergyPlus, the Window AC object is a piece of zone equipment that is relatively easy to define (see example file WindACAUTO.idf). For the Unitary System object, you need to define an air loop which means additional inputs that are probably not needed for what you are doing.

SET POINT TEMPERATURE

Are there any standards (or tables) for the set point and set point back of different rooms of a house during the day and night?

Answer

These kinds of standards are usually described for a country in that country's energy standards document. The ASHRAE Handbook of Fundamentals may also have some information. You can use the "comfort" reporting of EnergyPlus to determine whether people WOULD be comfortable at set back temperatures.

MULTIMODE HUMIDITY CONTROL ISSUES

I am trying to model a large unitary system with humidity control. No matter how I try, I cannot get the humidity control portion of the coil to work. Furthermore, I am told that the part-load ratio limits are exceeded. I am totally stumped. What I am trying to do is create a system that delivers air at a consistent temperature of 20C and a wetbulb of 12.5C. The only way I can see to do this is to create a dummy zone, with no loads, that serves as the controlling zone for the humidistat. Is there a better way to do this?

Answer

1. If you want the supply air to have a fixed humidity set point, then use Set Point Manager:Scheduled with the control variable set to MAXHUMRAT.
2. The multimode coil in this input has the same performance curves for both "Normal Mode" and "Dehumid Mode." Without different performance for the dehumid mode, the coil cannot do anything special. This option is intended for systems that switch refrigerant flow, such as sub-cool reheat, to achieve a lower SHR in the "dehumid" mode. If I understand correctly, the DX system you are modeling does not operate this way. I am guessing that it uses cool/reheat to achieve the desired supply air humidity. For that, set the DXSystem:AirLoop "Dehumidification Control Type" to CoolReheat.
3. The multimode coil is not necessary for this, but it won't hurt. The extra input will just be ignored.
4. And, finally, add the following report variables for the node leaving the DX coil so you can monitor the setpoints and results:

System Node Temp
System Node Setpoint Temp
System Node Humidity Ratio Max
System Node Humidity Ratio

FAN COIL UNIT WITH A FRESH AIR HANDLING UNIT

Will EnergyPlus model a fan coil system with a fresh air handling unit?

Answer

A fresh air handling unit that feeds directly into the outside air intake of the fan coil units is not currently supported. However, you can approximate this fairly well by having the pre-treated outside air feed directly into the zone and the fan coil unit will serve the net remaining load. Here's how to do this:

Define an AIR PRIMARY LOOP to treat the fresh air and use DIRECT AIR terminal units to feed the zone. In the ZONE EQUIPMENT LIST, set the heating and cooling priorities to 1 for the DIRECT AIR (fresh air system) and 2 for the fan coils.

Use a NODE LIST object to specify more than one inlet node for the zone.

The fresh air system will need a return air node and a return air path, even if there will not be any return flow, it must be defined.

AIRFLOW NETWORK MODEL

In the documentation it says that the AirflowNetwork model can calculate multizone airflows driven by wind when the HVAC system fan is off or when no forced air distribution system is specified for the simulation. Yet, I've read the Engineering Reference document for those topics related to advanced calculations of Natural Ventilation, (i.e., AirflowNetwork model considering no HVAC system defined), and it states that I need to "simulate SYSTEM" or "predict SYSTEM capability", etc. How, then, does the process of prediction-correction works if there's no "system to simulate"?

Answer

The documentation is correct. The AirflowNetwork model can calculate multizone airflows driven by wind, either the HVAC system fan is off with inputs of forced air distribution, or no forced air distribution system is specified. In addition, the model can calculate multizone airflows driven by wind and forced air through the forced air

distribution system when the HVAC system fan is on. In other words, the model calculates multizone airflows with wind only or with wind and forced air together.

Natural ventilation is a way to deal with zoned airflows without including an air distribution system. The alternative way is to use the AirflowNetwork model to calculate multizone airflows driven by wind with the similar approach, excluding air distribution system.

EnergyPlus always calls Predictor and Corrector with and without HVAC systems. The difference is that if there is no HVAC system, the system load in the corrector is zero.

PURCHASED AND OUTSIDE AIR

I'm using purchased air to determine annual thermal and cooling loads. And I'm using the infiltration method to handle the constant supplied outside air. Does the infiltration consider the latent gain of the outside air?

Answer

Yes, infiltration considers latent gain/loss.

Question

I also tried to handle the outside air as part of the purchased:air and not use infiltration. However, whenever the flow drops below the specified outside air flow rate, the system generates an error. How can I solve this?

Answer

You cannot solve this. That is why the warning is there. Purchased air is a variable volume device. There will always be a low-load condition at which the flow required to meet the zone load is smaller than the required outside air flow. You must use INFILTRATION or VENTILATION to add the outside air load to the zone when using PURCHASED AIR for an annual simulation.

GENERATE A DXF DRAWING

I'm trying to build a model with EnergyPlus and have finished inputting the surface geometries. When I tried to create the DXF file using EP-Launch I got an error message. I checked my input file but found no errors, only some warnings. If I can't export the DXF file, does it mean my model is in error?

Answer

There are two ways to generate the dxf drawing:

Add "report, surfaces, dxf;" to the idf file and run an EnergyPlus simulation. An output file with .dxf extension will be created. This can then be opened using the "Drawing Files" output button, or the DXF button on the quick-open panel in EP-Launch.

If the dxf file does not exist (or is older than the idf file), pressing the "Drawing Files" button in EP-Launch will cause WinEPDraw to run and create the dxf output file, and then it will open.

You must have a DXF viewer installed on your computer in order for EP-Launch to open the dxf file. If you do not have one, you can download VoloView Express (an old Autodesk program that is no longer supported) at <http://www.gard.com/vve201setup.exe>. After the viewer is installed, you must use Autofind or Select Drawing Viewer in the file menu in EP-Launch to tell it what program to use for dxf files. That should solve the problem.

PARTITIONS AND VOLUME

When an object is modeled as Heat Transfer:Internal Mass, its thermal mass is taken into account in the simulation. But is its volume subtracted from the zone volume in which the object is located? In the case of partitioning, this volume can be negligible but if furniture is being modeled (say filled bookshelves) would EnergyPlus subtract the volume occupied by all the furniture from the zone's total volume? In some cases, the volume occupied by these objects can be significant. I am curious to know if this is taken into account or not by EnergyPlus.

Answer

The zone volume that is referred to in various parts of the outputs is not mass, but dimensional. Therefore, it refers to the surfaces of the zone. Internal mass neither adds nor subtracts from the Zone Volume. It does, however, represent mass / thermal mass that should be taken into account during the calculation of the heat balance.

"AIR" PARTITIONS

How do I describe the construction of a virtual wall between perimeter zone and interior zone? Should Material:Air be applied? And if so, can light and heat go through the air wall partition? Can I set the thickness of the air wall?

Answer

This is a difficult feature to model. Material:Air will not transmit light, it is simply a resistive layer. A window subsurface is required to transmit solar and visible light from the perimeter zone to the interior zone. If the two zones will generally be controlled to the same temperature, then air exchange between the perimeter and interior can be neglected. If a significant temperature difference is expected, then MIXING, CROSS MIXING, or AIRFLOWNETWORK:* objects are required to model the air exchange. There is currently no method for modeling radiant exchange between surfaces in different zones.

WINDOW AIR CONDITIONER NODES

I need to simulate a room with a window air conditioner and I am having problems setting the air nodes. Is there an example? The written explanation is there but I didn't understand it very well. Is the scheme for a window air conditioner is similar to that shown in Fig. 87 in the Input/Output Reference (this figure is related to packaged terminal heat pump)?

Answer

Yes, Figure 87 for packaged terminal heat pumps is similar to the window air conditioner, except you would omit the DX heating coil and supplemental heating coil.

To see how to set the air nodes, look at the example file "WindACAuto.idf."

ROOM AIR NODE TEMPERATURES IN MUNDT MODEL

I am sizing a cooling system for a 7-meter tall industrial building by simply using a purchased air system and the Mundt model for a displacement ventilation air pattern.

I found that the return air temperature (Mundt Room Air Model Air Node Temp) and the Zone Mean Air Temperature are always about the same, i.e., close to the value of the Cooling Setpoint Temperature, despite the air temperature near the floor being slightly above the Cooling Supply Air Temperature ($T-20$ C). Could anyone help me understand this?

Answer

The Mundt model is a relatively simple model that only predicts a different temperature for the floor air "node." The single-slope air temperature profile from the Mundt model doesn't work well for the top portion of the zone so the other air temperatures are basically the zone mean. The 7-meter height of the building is probably too high a space for the Mundt model and, unfortunately, for the present there is no other room air model in EnergyPlus that would be better. I would suggest assuming a vertical air temperature profile for the space and then modeling the implications using the User Defined Distribution Controls.

DIRECT OUTSIDE-AIR SYSTEM

Any hints on how to use EnergyPlus to model a Direct Outside Air (DOAS) system?

Answer

EnergyPlus can model DOAS by using an AIR PRIMARY LOOP with DIRECT AIR terminal units to deliver pre-conditioned outside air directly into one or more zones. Then a zone equipment system, such as fan coil or PTAC or PTHP, is used to meet the net zone load. Be sure to list the DIRECT AIR units with priority = 1 in the ZONE EQUIPMENT LIST. Unfortunately, there is no example file (yet).

UNIT CONVERSION

Any hints in using ConvertESOMTR to change joules into kWh in the eplusout.eso files ?

Answer

The instructions for the use of ConvertESOMTR are in the Auxiliary Programs document on page 175 (pdf page 187). To use ConvertESOMTR use the option under the view menu.

LOADS NOT MET / HOURS NOT MET

Is it possible to report hours in the "load not met report" for the whole building, instead of each individual room?

Answer

No, but that would be a useful new report variable.

You might find the ASHRAE 55 comfort warnings to be useful for this purpose. In the PEOPLE objects, set "Enable ASHRAE 55 comfort warnings" to Yes. This enables new report variables and also generates a warning in the error file whenever the number of hours uncomfortable exceeds 4% of the simulation hours. For more details, search for "ASHRAE 55" in the Input Output/Reference.

SHADING DEVICES

I want to simulate a cross-ventilated zone having different types of shading devices (19 types derived from two main devices on the exterior wall). Some are perpendicular to the wall, and some are parallel to the wall. I want to see the effects of these devices on the cooling loads. I modeled all these devices using CAD. When I import my drawings, will the program be able to recognize the information?

Answer

There is no current auto import of CAD data. One or more of the private-market user interface tools can display the CAD drawing and allow you to trace over it. Shading devices are modeled three ways in EnergyPlus: as a shade on the window surface, as a blind (movable slats) on the window surface, or as a collection of exterior shading surfaces (see SURFACE:SHADING:*).

Question

When the window is open, how do I define a shading device? In the examples, there were glazed surfaces.

Answer

In EnergyPlus, a window (SURFACE:HEATTRANSFER:SUB, type=WINDOW) is required to transmit solar gains into the building. Windows are also used as controllable openings for natural ventilation (see AIRFLOWNETWORK). The solar transmittance of the window area does not change depending on the open or closed state of the window. If the window is never glazed, but is simply an opening in the exterior wall, then define the glazing properties to be nearly perfect (e.g., transmittance = 0.99999, reflectances = 0.00001).

Question

When the window is open, I want to consider the impact of both shaded and unshaded areas on the floor. Will this information import into EnergyPlus?

Answer

EnergyPlus will determine the net direct and diffuse solar radiation, which passes through the window, accounting for all shading devices. If you specify, Solar Distribution=FullInteriorAndExterior or FullInteriorAndExteriorWithReflections in the BUILDING object, EnergyPlus will determine on which surfaces the incoming direct solar falls and account for this in the heat balance for each surface. The sunlit area and sunlit fraction can be reported for exterior surfaces, but not for interior surfaces.

Question

Can Energyplus calculate the surface temperatures?

Answer

Yes, both inside and outside surface temperatures can be reported.

WINDOW REVEALS

I've read through all the documentation on window reveals, and I'm still confused. According to the documentation, the window reveals are specified from the inside reveal edge → glass, and EnergyPlus figures out the Glass → outside reveal. In other words, the input specifies the glass distance from the inner wall surface, and EnergyPlus calculates the glass distance from the outer wall surface.

However, as far as I know, EnergyPlus knows nothing about wall thickness (1-D calculations), so on what basis does it calculate the window-to-outer-wall surface. And will the default setting of "0" place the window on the inside wall surface (full depth reveal) instead of the U.S. architectural norm of the outside wall surface?

Finally, why does EnergyPlus take the inner reveal as the augment? The inner reveal has no impact on shading (and thus solar gain, although it does impact daylight diffusion), whereas the exterior reveal does.

Answer

Yes, this can be confusing.

Surfaces and SubSurfaces in EnergyPlus have no thickness in and of themselves. It is only when the vertex coordinates for a Window SubSurface place it in a different plane from its host Wall Surface, presumably some distance from the "inside" face of the Wall Surface plane, that an outside reveal is detected by EnergyPlus. Inside reveal depths and absorptances (for sill and other edges) are input using the WindowFrameAndDivider input object.

As to the impact of these reveals on solar gains, read the details in the Input/Output Reference under WindowFrameAndDivider/Reveal Surface Fields, in particular the following:

"The net effect of beam solar reflected from **outside** reveal surfaces is to increase the heat gain to the zone, whereas the effect of beam solar reflected from **inside** reveal surfaces is to decrease the heat gain to the zone since part of this reflected solar is transmitted back out the window."

OUTSIDE REVEAL SHADOWING

I want to see the solar gain effect of placing a window in either the outer or inner surface of a wall with 0.45 m thickness. The documentation states that this is related to the outside reveal depth (ORD), which is calculated from the inside reveal depth (IRD) that we input and from the wall thickness (WT) and glazing thickness (GT). My understanding is that $ORD = WT - IRD - GT$, since ORD and IRD are measured from the outside and inside glazing surfaces until the respective wall surfaces.

So, in order to change the glazing position between the inner and outer wall surfaces we should change the IRD value. The problem is that changing this value from its default (0) caused EnergyPlus to crash. By default, doesn't EnergyPlus consider the glazing in the outer surface?

Answer

The outside reveal depth is defined by the coordinates of the window vertices relative to the plane of the base wall surface. For example, if the corners of a south-facing wall are at $y=10$, then window with a reveal of 0.2m would have coordinates with $y=10.2$. This is discussed on p. 179 (pdf p.217) of the Input/Output Reference. The inside reveal must be defined using the window frame and divider inputs. Note that the material thicknesses are not known by the solar shading algorithms.

ZONE WITH NO WALLS

How do you model zones that do not have a wall between them? Should I leave the imaginary surface out, or should I define it using a wall with a full size window and miniscule heat transfer properties? If there is significant daylighting, then defining the imaginary surface a wall would block this out.

Answer

Unfortunately, there is no good solution to modeling this case. Here is a (rather) long answer in which some of the issues are examined.

It is extremely difficult to model the interactions between thermal zones that are connected by a large opening. If the zones are controlled to the same conditions, then there is little to be gained by making them interact, so you could neglect any connections between the zones. In fact, if this is the case, you might consider combining the spaces into a single thermal zone. If you expect the zones to have significantly different temperatures and/or humidities, then use one of the following options. If they are modeled as separate zones, EnergyPlus models only what is explicitly described in the input file, so simply leaving a void (no surfaces) between two zones will accomplish nothing -- the two zones will not be connected. The main interactions which occur across the dividing line between two zones that are fully open to each other are these:

1. Convection or airflow, which will transfer both sensible heat and moisture. Some modelers use MIXING (one-way flow) or CROSS MIXING (two-way flow) to move air between the zones, but the user must specify airflow rates and schedules for this flow, and it cannot be automatically linked to HVAC system operation. Other modelers use AirFlowNetwork with large vertical openings between the zones as well as other openings and cracks in the exterior envelope to provide the driving forces. It can also be connected with the HVAC system (for limited system types). This requires a much higher level of detailed input and should be used only if the detailed specification data is available. If the two zones are controlled to similar conditions, this effect could be safely neglected.
2. Solar gains and daylighting. The only way to pass solar and daylight from one zone to the next is through a window or glass door described as a subsurface (Surface:HeatTransfer:Sub) on an interzone wall surface. Note that all solar is diffuse after passing through an interior window.
3. Radiant (long-wave thermal) transfer. There is currently no direct radiant exchange between surfaces in different thermal zones. Windows in EnergyPlus are opaque, in order to direct radiant exchange, so an interzone window will not behave any differently than an opaque interzone surface for this aspect. However, a large interzone surface (opaque or window) would result in some indirect radiant exchange since the interzone surface will exchange directly with surfaces in zone A and in zone B. The surface thermal resistance should be low in order to most closely approximate this effect.
4. Conduction. If an interzone surface is placed between the two zones, it will conduct sensible heat between the two zones. Using a low thermal resistance helps to move radiant exchange between the zones.
5. Visible and thermal radiant output from internal gains. These gains will not cross zone boundaries. But again, they will impact any interzone surfaces, so some of the energy may move across to the next zone.

Reply

Great answer. This information should be saved and linked to the EnergyPlus help files.

UNENTERED OTHER ZONE SURFACE

If "UnenteredOtherZoneSurface" is selected as the Outside Face Environment for a particular surface, does EnergyPlus determine the proper area from the corresponding surface of the specified ZONE, or should I make sure that the specified ZONE has a surface with area equal to the subject surface?

Answer

When using OtherZoneSurface, you must describe the surface twice, once in each zone. For an example of UnenteredOtherZoneSurface, see example file 5ZoneAirCooledConvCoef.idf.

CORRIDOR MODELING

I am doing a simulation that includes a corridor (a zone). The corridor is the outside surface zone for several offices (each one a different zone). Do I need to fragment my corridor wall so it coincides with the offices walls (so surfaces on the offices correspond to surfaces on the corridor)? Or should I define the surfaces in the offices using the wall on the corridor as their outside zone and then only define one wall on the corridor with "unenteredOtherZoneSurface"?

Answer

A new feature in EnergyPlus 2.0 will automatically define one wall, which is in contact with the corridor, for each office. Here's how:

1. In each office describe the wall next to the corridor using "unenteredOtherZoneSurface" and specify the Corridor as the outside face environment object.
2. Do not describe any of these walls in the corridor zone; the "unenteredOtherZoneSurface" option will create them internally. You can see them in output reports, but you do not need to describe them in the idf file.

EPW FILES

In the .epw files, which columns are the ones for the temperature of the site and which are for direct radiation?

Answer

Note that the data in the EPW files is comma-delimited, not "columns."
The format of the EPW file is described in AuxiliaryPrograms.pdf , beginning on page 37 (pdf page number 49), "For those interested in creating their own weather data in CSV or EPW formats or reading the .csv or .epw files that are produced by the Weather Converter program....."

WINDOW WITH SWITCHABLE GLAZING

Can EnergyPlus simulate Window construction tinted automatically with solar incident?

Answer

Look under WindowShadingControl. You can choose SwitchableGlazing as the Shading Type and then select from several triggers for darkening the glazing.

Question

Do the triggers mean adding shade behind the window glass?

Answer

By trigger, I mean a control variable that sets a threshold for darkening the tint of electrochromic glazing. This is explained under WindowShadingControl Shading Control Type.

You did not ask about shades in your original message. Note that you can also use the Shading Control Types in WindowShadingControl to deploy (add) a shade that has been defined as inside, outside, or between glass layers of a window. You will have to read the Input/Output Reference documentation on Material:WindowShade.

INTERIOR BLINDS

In a model I made to simulate heat transfer from the envelope, I made runs with and without interior blinds for windows. I thought that the cooling loads through the envelope due to solar heat gain should be higher without blinds than with blinds, but the results showed that sometimes the results were higher with the blinds (early morning on the east facade and late afternoon on the west façade). I also tried interior shade, but the results are similar.

Answer

Placing blinds and/or shades on the interior of a window will not stop solar gains from entering the zone. Solar will still be transmitted through the window and will warm up the blind/shade surface, which is inside the zone. Some solar will be reflected back out the window, but this will not reliably lower the overall solar gains relative to a window without an interior blind or shade.

Try placing the blind/shade on the exterior of the window for comparison.

CONVERT DOE-2 FMT TO ENERGYPLUS EPW

How do I convert the DOE-2 FMT to EnergyPlus epw?

Answer

The WeatherConverter will do that for you. DOE-2 FMT (text format) is one of the supported formats. From the Start button → Programs → EnergyPlus V2-0 Programs → click on Weather Statistics and Conversions. More information is also provided in the Auxiliary Programs PDF.

AIRFLOW OBJECT -- DESIGN VOLUME FLOW RATE

I want to simulate natural ventilation and have a question about the Airflow Object. How do I explicitly define the design volume flow rate?

Answer

When using INFILTRATION and VENTILATION (simple airflow method), you must calculate the design volume flow rate using other software, handbook formulas, or "rules-of-thumb."

You can also use the AIRFLOWNETWORK simulation to compute flow rates using descriptions of openings, flow coefficients, and wind pressures. See AIRFLOWNETWORK SIMULATION, and AIRFLOWNETWORK:MULTIZONE:*

MODEL CREATION AND AUTOSIZING

I have a basic question about model creation using EnergyPlus. We used the autosizing function to determine several factors for a cooling tower and detailed cooling coil but we ended up with a very unstable model. Is it typical to place autosizing parameters in the unknown components in EnergyPlus, run the appropriate sizing parameters, and then replace the autosize with the resulting values? In what cases will the "design" parameters (cooling tower W/k, fan flow, cooling coil capacitance etc.) not allow stable operation for all types of weather?

Also, I have been getting errors about an "unacceptable loop demand range" causing cooling tower shutdown for various errors and that the "condenser loop volume is too small for stable operation." The values that appear to cause the errors are the actual values observed in the building.

Answer

Generally, a fully autosized run behaves very well, and it is not typical to go back and replace autosize with hard values.

The "unacceptable loop demand range" may just be an input issue. In the range-based operation objects, the last upper range limit should always be a very large value to avoid an early iteration landing in the out-of-range zone.

The "condenser loop volume is too small for stable operation" appears to be happening at odd times (might be a small bug) but it should be adjusting this upward automatically and running fine.

Question

I understand that it may not be necessary to remove the autosizing parameters; however, the run time is excessive when using the system, plant, and zone sizing in the run parameters.

Is there a recommended procedure for implementing values, in place of the autosizing numbers, that would allow me to run the model without these run control features active? It seems that I should be able to "size" the plant once, based on typical equipment parameters for a building of some cooling load, and keep those parameters for each day of the year. Is there any critical limiting factors that would keep me from setting the coil and cooling tower characteristics constant throughout the simulation year?

Answer

I am surprised that autosizing would add significantly to run time. Sizing performs a loads-only (purchased air simulation) for each DesignDay, then calls the systems once-through to size components.

All of the computed sizes are reported in the eio output file and once these are determined, they are fixed for the entire simulation -- nothing is changing over the course of the year.

The procedure for using hard sizes would be to take the values reported in the eio output component sizing reports and copy/paste them into the idf file, replacing each instance of "autosize." I doubt that you will find much difference in run-time between the autosized and hard-sized versions of the input, but if you do, please send us the input files for further investigation.

AUTOSIZING MULTIPLE CHILLERS

My model that includes two chillers and I used design objects to autosize them. However, I found that only one of the chillers works!

Answer

EnergyPlus autosizing does not understand multiple pieces of equipment. It will size each piece to meet the full load, in this case both chillers. Whenever you use multiple boilers, chillers, or cooling towers, follow this procedure:

1. Run the simulation with the plant equipment autosized.
2. Find the reported equipment capacities (for chillers, boilers, etc.) in the eio output file. The capacity of any one chiller or boiler will be the full-load capacity. (You should see the same capacity reported for each chiller, for example.)
3. Make a new version of the input file with hard sizes specified as needed to allocate the full-load capacity among the chillers (or boilers, etc.).

AUTOSIZING AND PARAMETRICS

I just finished the simulation of a baseline building using EnergyPlus. All of the equipment in this building was autosized. I plan to model multiple energy conservation measures using this baseline building. However, I am concerned that, with each simulation run, when I model an energy conservation measure (and add it to the baseline model), the capacity of the building equipment (boiler, chiller, fan, etc.) will change. Do I have to hard-size the equipment (boiler, chiller, fan, etc.), based on the values from the baseline model's EIO for the runs where I model these measures? Will the autosized capacity of the equipment to change with each run? Or, should I expect the autosized values to remain the same with each run?

Answer

If the energy conservation measures affect the building's zone loads, then the autosized zone design flow rates will change and, downstream, the boiler, chiller, fan, etc. An example would be a different type of window -- that would affect the autosizing results.

Question

Thanks for the response. Do you know of an easy way to input the autosized values from the EIO into the IDF? Inputting these values into the IDF manually seems time consuming and I am looking for an easier way.

Answer (work-around)

There is no automated way to do that. However, you could try this: Specify fixed flow rates in the ZONE SIZING and SYSTEM SIZING objects using the flow/zone and flow/system options. Then let the HVAC components autosize. The fixed flow rates will decouple the autosizing from actual zone loads, and the rest of the system sizing is driven by the supply air flow rates. I can't remember for sure if the zone sizing factor is applied to these values (it is for some inputs, not for others), so set the sizing factor to 1.0 for this approach.

WIND SPEED FOR SURFACE ORIENTATION

How does EnergyPlus determine the "Surface Ext Wind Speed" output from site wind speed, as listed in a design day object? There must be a formula that involves surface orientation but I cannot find it in the documentation.

Answer

The simple calculation does not take orientation into account.

From the Input/Output Reference:

Surface Ext Wind Speed [m/s]

The outdoor wind speed is calculated at the height above ground of the surface center.

From the code:

```
! LocalWindSpeed = Windspeed [Met] * (Wind Boundary LayerThickness [Met]/Height  
[Met])**Wind Exponent[Met] &  
! * (Height above ground / Site Wind Boundary Layer Thickness) ** Site Wind  
Exponent  
! LocalWindSpeed = WindSpeed * WeatherFileWindModCoeff * (Z / SiteWindBLHeight) **  
SiteWindExp
```

The calculations are also discussed in some detail in the Output Details and Examples document (file: eplusout.eio)

SYSTEM VOLUME FOR CHILLED AND CONDENSER WATER

In my experience with using EnergyPlus, it becomes very unstable at system volumes under 10,000 gallons.

If I let the program autosize this value to something very large (say, 120,000 gallons) what keeps the program from modeling such systems like a thermal storage loop when the system volume is so much larger than the actual building? How can I improve program stability for reasonable system volumes?

Answer

There are example files that auto-calculate loop volumes much smaller than 10,000 gallons and show no stability problems. For example, the reported loop volume for SolarCollectorFlatPlateWater.idf is 0.63787 m³ (or 168) gallons. The auto-calculated loop volume is proportional to the loop maximum flow rate and the length of the zone time step.

If the loop maximum flow rate input is not autosized, be sure that it is not larger than it should be.

LOADS REPORTING

How do I get EnergyPlus results in the form of heating loads, cooling loads and lighting loads?

Answer

There are several different ways to report these. It depends on what exactly you are looking for: zone-level loads, equipment or coil-level loads, energy consumption by end-use, other?

For energy use by end-use, try this command:

```
Report:Table:Predefined,  
Annual Building Utility Performance Summary; !- ReportName1
```

Some example monthly report objects may be found in

```
C:\EnergyPlusV1-4-0\DataSets\StandardReports.idf
```

VIEW FACTORS ERROR

I'm modeling an attic space in an A-Frame building; the ceiling has two sloped sides and one flat side. I got the following error. Can you tell me if this error was based only on the geometry or should I start digging into the coordinates.

```
** Severe ** View factors not complete. Probably bad surface descriptions in  
zone="OPEN_LAB_404".  
** ~~~ ** Enforced reciprocity has tolerance (ideal is 0.0)=1.3595E-002
```

Answer

First, check REPORT, Surfaces, Details; and make sure that all five surfaces enclosing the attic have the correct azimuth and tilt (result go in the eio output file). If these are correct, you might try breaking the two large sloped

surfaces into two surfaces each. In extreme geometry situations, the view factor algorithm sometimes works better with several small surfaces rather than a few large ones.

SIMULATING AN EXTRA-LIGHT STRUCTURE

I need to simulate a very light envelope with 50% window-to-wall ratio (WWR); the walls are composed of three layers: metal, insulating material and again metal. Are there any restrictions to simulating such a structure with EnergyPlus? I am concerned, having such light envelope, that EnergyPlus might experience some numerical difficulties for convergence.

Answer

Describe the materials as regular materials (instead of just R values) and you will be better off. There is one example file that is described with very light (just R) values but it's not suggested for general use.

WIND PRESSURE COEFFICIENTS

There's something I don't completely understand about the wind pressure coefficients. In order to find the Cp values around my building, I've done CFD analyses. In my case, the predominant wind comes from the North; the wind that flows from North to South is what I've simulated using CFD so I know both the pressure distribution and the Cp values. What's the meaning of the 45° increments for each value of Cp?

Answer

Simulating the building with wind from the North gives you the Cp values for all surfaces at that wind angle (0). But this is just 1 Cp value for each surface. The wind in any simulation will come from many (or all) directions. If you simulate your building from the East, South, and West, you will then have the Cp values for all surfaces at wind angles of 90, 180, and 270. But, again, this is only 4 wind directions. If you simulate the building with wind from the North, NorthEast, East, SouthEast, South, SouthWest, West, and NorthWest, you will then have the Cp values for all surfaces at wind angles of 0, 45, 90, 135, 180, 225, 270, 315, and 360 (45 degree increments). Now you have Cp values for all surfaces at 8 wind directions. Get the idea? Since you already have the CFD model, this should be pretty easy.

CONVECTION COEFFICIENT

In my building model, I want to adjust the convection coefficient for the walls/floor/ceiling to the air by myself. Are there typical values for a room of about 16m³ and infiltrations of 27m³/h? And how does that value change if I introduce an HVAC with 10ACH?

Answer

You can report out the interior convection coefficients for a surface using the report variable called "Surface Int Convection Coeff."

First try reporting out the coefficients using the "Detailed" algorithm, which is an option under the INSIDE CONVECTION ALGORITHM object. The "Detailed" algorithm sets the coefficients based only on natural convection. Next, you can try the "CeilingDiffuser" algorithm, which adjusts the convection coefficients based on the ACH of air flow through the zone node. You will need to set up an air loop to provide 10 ACH. You should be able to take one of the example files and modify it for your needs. The one called FurnaceWithDXSystem.idf might be a good place to start. With the "CeilingDiffuser" algorithm, keep in mind that for ACH < 3.0 the correlation is not valid and the algorithm reverts to "Detailed." Finally, to set your own convection coefficients on a surface, use the CONVECTIONCOEFFICIENTS object.

The objects mentioned above are also documented in the Input/Output Reference and the Engineering Reference.

HOT/CHILLED WATER RESET SAVINGS

I am trying to estimate the energy savings of implementing a hot water reset as a function of outside air. When I implemented the hot water reset, the energy consumption of the boiler did not change – there were no energy savings. Why? I am using the BOILER:SIMPLE object; does it take into account the temperature of water delivered in its energy consumption calculation?

Answer

No, BOILER:SIMPLE does nothing more than apply the nominal efficiency and a part-load curve. It does not model any jacket losses, and so its efficiency is not sensitive to the supply temperature. Also, with no piping losses in the simulation, that effect is not present either.

So, I would expect the same boiler energy consumption for these two cases. I can't think of an easy way to model this with current capabilities.

Question

I am having the same issue with chilled water reset -- is this to be expected?

Answer

No. Higher chilled water supply temperatures should reduce chiller power consumption for the same load. This assumes that the chiller EIR as a function of temperature curve is valid. One would also expect the load to fall because the latent cooling at the coils should fall with higher water temperatures.

SOLAR OUTPUT VARIABLES

I have a question about the Solar Exterior Surface Incident output variables. EnergyPlus reports the units in $[W/m^2]$ if dimensions for each surface are known. Since the four points in the space must be defined, why not report directly in $[W]$? Then, the total solar radiation per area unit incident on a surface is the sum of Surface Ext Solar Incident + Surface Ext Solar Sky Diffuse Incident. Am I misunderstanding these output variables?

Answer

EnergyPlus does know the area of the surface, but the goal of these report variables was to report the solar intensity, which is easier to compare between surfaces when it has been normalized by the area. If incident solar for each surface in W would be useful, we can add it to the suggestion list. Most report variables are defined in the Input/Output Reference, just search for the variable name.

Here are four related report variables for incident solar:

Surface Ext Solar Incident

Surface Ext Solar Beam Incident

Surface Ext Solar Sky Diffuse Incident

Surface Ext Solar Ground Diffuse Incident

The first one is the sum of the last three.

SENSIBLE HEATING AND COOLING RATE FOR ONE ZONE

From the results of one of my simulations, I found that during some time steps for one zone the sensible heating and sensible cooling rate were both greater than zero. Can you help me understand this?

Answer

It is most unlikely that this would happen during "detailed" reporting; however, it is quite likely during time step (i.e. Zone Time Step) or hourly reporting. The HVAC time step automatically reduces itself below the zone time step to get to convergence. Read about the Time Step in Hour and System Convergence limits as well as searching the documentation for "time step (timestep)."

GROUND TEMPERATURES AND THE SLAB PROGRAM

My question is about calculating ground temperatures. After I reviewed the Auxiliary Programs documentation, I tried to calculate ground temperatures, but when I ran the slab program, I got error messages.

Answer

The Slab program needs its own IDF file, written to the specifications shown in the Auxiliary Programs document. From the example you submitted, it would appear that you pointed the IDF file at an EnergyPlus IDF file instead. SlabExample.idf in the Ground Temp Folder should give you a good starting point.

INCREASING HEAT TRANSFER WITH INDUCED TURBULENCE

I added ribs to the surface of a wall in order to induce turbulence. Do you have any ideas about how to model the changes to heat transfer to walls, etc.?

Answer

It all comes down to convection coefficients. In EnergyPlus, you can report convection coefficients (Surface Int Convection Coeff and Surface Ext Convection Coeff) and you can control them with the ConvectionCoefficients object or by simply changing the Roughness field in the outside material layer. So, even without knowing the real relationship between ribs and heat transfer, you could get an idea of the potential impact of, say, a 10-fold increase in convection coefficients.

INSIDE CONVECTION COEFFICIENT

I would like to define my own values of inside convection coefficients in the Energyplus idf file. How?

Answer

Look at example file 5ZoneAirCooledConvCoef.idf and the "ConvectionCoefficients" object. Also, look in the Input/Output Reference for the same object and the ConvectionCoefficients:MultipleSurface object descriptions. These will allow you to change convection coefficients for one or many surfaces in your file.

THERMAL COMFORT AND THERMOSTATIC

I need to control Thermal Comfort during office hours and then switch over to Thermostatic control during non-office hours (page 847 / pdf 855) of the Input/Output Reference Manual. This is the error I'm getting:

```
** Severe ** GetZoneAirSetpoints:  
** Could not find Dual Thermal Comfort Setpoint with Deadband  
** Schedule:Fanger=TEST
```

TEST is the name I gave to Thermal Comfort Control Type Name #1 in the Zone Control:Thermal Comfort object. Do I enter TEST in the Setpoint Manager? I already have a Schedule type (PMV, range -4:4, Continuous Numeric) DaySchedule (Thermal Comfort Day, PMV, with hourly values of 4 and 0) WeekSchedule (Thermal

Comfort Week, Thermal Comfort Day for all fields Schedule (Thermal Comfort, PMV, Thermal Comfort Week, 1/1 to 12/31)?

Answer

The reason you got the error is that you don't have an object of Dual Thermal Comfort Setpoint with Deadband:Fanger with object name = Test. After defining the object, you may need two setpoint PMV schedules. The schedule you mentioned looks like thermal comfort type schedule, which allows you to change comfort type every time step. Please look at an example file of FurnaceWithDXSystemComfortControl.idf, which may give you a good starting point.

```
!- ===== ALL OBJECTS IN CLASS: DUAL THERMAL COMFORT SETPOINT WITH DEADBAND:FANGER =  
DUAL THERMAL COMFORT SETPOINT WITH DEADBAND:FANGER, Test,  
!- Name Heating PMV Setpoints,  
!- Fanger Thermal Comfort Heating Setpoint PMV SCHEDULE Name Cooling PMV Setpoints; !- Fanger  
Thermal Comfort Cooling Setpoint PMV SCHEDULE Name
```

SUPPLY FAN VARIABLES

I am doing a building simulation in which compact HVAC is used. In the field for supply fan operation mode I have chosen cycling (not continuous). How do I find the power consumed by the supply fan separately (cooling and/or heating)?

Answer

Use the Fan Electric Power and Fan Electric Consumption report variables. These variables are described in the Input/Output Reference document for the Fan:Simple:OnOff object. If you want to know how much fan energy is consumed in cooling mode and how much fan energy is consumed in heating mode, then you will have to use a spreadsheet to perform the calculations. Report the fan cooling and heating energy. Then in the spreadsheet, sum the fan energy when cooling energy is greater than 0. Do the same for the fan when heating energy is greater than 0.

ZONE CONTROLS AND THERMOSTATS

I want to calculate "zone sensible load predicted," "zone sensible load to heating setpoint predicted," and "zone sensible load to cooling setpoint predicted" against the setpoint. I entered a value in the "zone controls and thermostats" object but the resulting value was different from the "input/output reference" statement. The "Zone sensible load predicted" results are all 0, "zone sensible load to heating setpoint predicted" results are all negative values, and "zone sensible load to cooling setpoint predicted" results are all positive values.

Answer

Zone sensible load predicted gives you the thermostat load. If it is zero, there is no load. The zone temperature is below the cooling setpoint and/or above the heating setpoint. This is the thermostat dead band region. The zone sensible load to heating setpoint predicted is negative because the zone temperature is above the heating setpoint. The zone sensible load to cooling setpoint predicted is positive because the zone temperature is below the cooling setpoint.

Change your thermostat setpoints and you will see that when there is a cooling load, the zone sensible load to cooling setpoint predicted will go negative when the zone temperature is above the cooling setpoint. And when there is a heating load, the zone sensible load to heating setpoint will go positive when the zone temperature is below the heating setpoint. The zone sensible load predicted should also show these same results.

If your thermostat setpoints are correct, you will have to change the zone loads. More outside air will likely cause a load in the zone for both winter and summer. Or adding equipment/lighting loads will cause a cooling load in summer (and possibly winter).

EVAPCOOLER RDD SPECIAL COOLER

I am using the RDDSpecial object to simulate an indirect evaporative cooler where the secondary air is building exhaust air. I want the object to behave like a wet coil indirect evaporative cooler. However, I can't figure out what the inputs for the object should be to make it behave like a wet coil, as opposed to a regular indirect pad cooler. I know there is an example file, but I don't know what type of cooler is in the example file. What should I input for the "Cooler Maximum Effectiveness"? What about the "Dewpoint Effectiveness Factor"?

Answer

The RDDSpecial model is fairly generic and can represent any type of indirect cooler with a constant wetbulb effectiveness. The words "effectiveness" and "efficiency" are somewhat interchangeable here. The value for "cooler maximum effectiveness" would be the same as what you use for "coil maximum efficiency" in the input for EvapCooler:InDirect:WetCoil. The Dewpoint effectiveness factor can be set to 1.0 -- it only comes into play for wetbulb effectiveness values well over 1.0 (as for M-cycle coolers).

FLOOR GEOMETRY

I have an L-shaped floor that shows up as a "polyline" with some thickness in DXF if I enter the 6 coordinates in one surface object. However, if I enter it as 2 rectangular surfaces they show up as 2 "surfaces" in the DXF file. Does the DXF rendering give any indication of how the surface is treated by EnergyPlus for load calculations?

Answer

Polyline/thick line is the default for a greater-than-4 sided surface representation in the DXF file. You can also choose to have the program "triangulate" that greater-than-4 sided surface for better pictures. But the inside representation/calculation has nothing to do with the DXF view.

HEAT PUMP DATA

I am simulating a room with a unitary heat pump, wherein I am trying to define the cooling and heating powers -- but don't know where to define them!

For example:

Cooling power=3500W

Heating power=3700W

Absorbed power during the maximum performance=1090W.

Answer

Look in the Input/ Output Reference document (pdf) and search for "UnitarySystem:HeatPump:AirToAir". For this system, you need to define an associated direct expansion (DX) cooling coil

[Coil:DX:CoolingBypassFactorEmpirical or Coil:DX:CoolingHeatExchanger Assisted] and a DX heating coil

[Coil:DX:HeatingEmpirical].

The performance of the cooling and heating coils is defined in these coil objects.

RESIDENTIAL HEATER SIZING AND CYCLING TIME STEP

I need to model a standard constant volume residential forced-air heater in EnergyPlus and see the impact of Sizing on efficiency and heater cycling. However, I haven't been able to figure out how to model the heater appropriately to make sure that it has a sufficiently small time step resolution so it can cycle and the difference in efficiency can be noted. My understanding is that the HVAC system time step is decoupled from the 15 minute building time step, allowing it to free-run with an adaptive time step (down to seconds if need be). Is this true? And, if not, is there anything that needs to be done to enable this behavior? Is there any way to see the high-res HVAC transients in the output?

Also, what is the best way to get a constant volume, constant BTU heater to model residential forced air systems?

Answer

You might want to have a look at the example file: FurnacePLRHeatingCoil.idf; it uses a Furnace:BlowThrough:HeatOnly system, which uses a gas heating coil (Coil:Gas:Heating).

By default, the HVAC time step is automatically reduced below the building (zone) time step as needed, down to a minimum of 1 minute if needed (not seconds).

See the 'Time Step In Hour' and 'System Convergence Limits' objects in the Input/Output Reference to understand the details.

COOLING COIL ERROR

I am getting the following error on a coil:water:cooling

```
** Warning ** SizeWaterCoil: Coil="EXTERIOR ZONES COOLING COIL",
** Cooling Coil has leaving humidity ratio > entering humidity ratio.
** will be sized for 50% of entering humidity ratio.
** Warning ** SizeWaterCoil: Coil="EXTERIOR ZONES COOLING COIL",
** Cooling Coil has part wet, part dry conditions.
** will be sized for 50% of design entering humidity ratio.
** Warning ** SizeWaterCoil: Coil="OPERATING ZONE COOLING COIL",
** Cooling Coil has leaving humidity ratio > entering humidity ratio.
** will be sized for 50% of entering humidity ratio.
** Warning ** SizeWaterCoil: Coil="OPERATING ZONE COOLING COIL",
** Cooling Coil has part wet, part dry conditions.
** will be sized for 50% of design entering humidity ratio.
```

I don't understand what the error means. In terms of the leaving humidity ratio, are we talking about the system design humidity ratio? This error is destroying the performance of the cooling coil.

Answer

This is coming from the coil autosizing calculation. The coil entering conditions are the mixed air conditions: recirculated air (if any) mixed with outside air. The outlet conditions are what you told the program in your System Sizing input as your desired design cooling supply air conditions. So the message is saying that the entering moisture content is less than the leaving moisture content.

Since a coil can't add moisture, the choice is made to try to remove half of the entering moisture.

HEATER ELEMENT EXPONENT

I need to model a simple heating system, including boiler, pump and radiator (baseboard). However, I didn't find an option to enter the heater element exponent (emission index) in the Baseboard Heater:Water:Convective object. How do I represent the non-linear properties of a radiator?

Answer

The baseboard heater in EnergyPlus, BASEBOARD HEATER:WATER:CONVECTIVE, is 100% convective. The equation in the Input/Output Reference, which defines the heat output as a function of UA, is incorrect. And, unfortunately, there is no section in the Engineering Reference for this model. The revised description for the UA field follows:

Field: UA

The overall heat transfer coefficient for the baseboard heater in watts per degree Celsius at design conditions. The UA is used in an NTU -- effectiveness calculation. An estimate of the UA can be obtained from:

$$q = UA \times (T_{\text{water,avg}} - T_{\text{air,avg}})$$

where q is the heat transferred from water to the air in watts; $T_{\text{water,avg}}$ is the average water temperature in °C; and $T_{\text{air,avg}}$ is the average air temperature in °C. Or the LMTD temperature difference can be used.

So, for convective heat transfer, the non-linear output of the heater as a function of water and air temperature should be accounted for by the model. But there is no radiant component, so that effect is lost. For the Fall 2007 release, there will be a new radiator model, which will be a combined convection-radiation model.

Currently, a true radiator can be approximated by using LOW TEMP RADIANT SYSTEM:HYDRONIC on a small surface.

ZERO AIR LOOP MASS FLOW RATE

I'm simulating a one-zone model to get familiar with Energy Plus. I constructed a simple Air Loop. The primary air system side contains Fan and coil:water:simple:heating elements, the zone equipment side contains a single duct:vav:noreheat element. I compared my file several times with example files, but I have still no mass flow rate in the Air Loop. I don't get any warnings, thus it seems to be a set point or controller fault.

Answer

In the BRANCH object "Branch_Air_supply_in" the maximum flow rate is set to zero (Version 2.0 generates a warning message for this condition, so you should upgrade). When specifying hard values for sizes (instead of autosize), the maximum (or design) flow rate in the AIR PRIMARY LOOP, BRANCH, FAN, and other components on the air loop main branch must all have the same value. And the sum of the terminal unit design flow rates must equal the main air loop flow rate.

WET BULB TEMPERATURE INCONSISTENCY

I noticed a difference in the wet bulb temperature in the eso file output, when compared to the wet bulb temperature found in the .epw file.

Answer

First, note that the epw file does not have a wet bulb temperature, it has a dew point temperature.

There is also the issue of interpolation. Perhaps if you ran Time Step in Hour = 1, you would see the temperatures exactly as they are on the epw file (hourly). At other time step frequencies, they are interpolated between the time steps to make a continuous progression.

CFM/TON LIMITS

I am having a problem getting PTAC cooling coils to size correctly. The coil cooling capacities are sizing smaller than they should (considerably smaller, by a factor of almost 6). The eio file reports that the coils are sizing right up at the limit of 450 cfm/ton (6.04E-05 m³/W). I am interpreting this to mean that the low tonnage is correspondingly limiting the air flow. However, I need more tonnage to supply zone temperatures. Why is the sizing routing limiting my tonnage? And how do I move them from this limit? The PTAC air flows are all autosized. My zone sizing design cooling temperature is 14C, which I believe to be fairly reasonable.

Answer

The air flow is calculated first and is fundamental; it is not altered. The cooling capacity is then calculated and is limited by 450 cfm/ton. The cooling flow rate is basically established by the zone design sensible cooling load, which is set by your design day weather, internal loads, outside air requirements, window-and-wall heat gains, etc. So, if you are seeing an error by a factor of 6, the questions would be these:

1. Are your design calculation inputs correct (does the design flow rate seem reasonable)?
2. Do you have a huge latent load (from internal sources or outside air)?

DAY-OF-THE-WEEK START DAY

For the RUNPERIOD, what determines the "Day-of-the-Week start day"? Is this day determined from the weather file, or can the day be chosen as I desire?

Answer

In general, the weather files used for energy analysis do not represent actual "years." If you look at some of the EPW files, you will see the actual years that were used to make up the data listed. Usually a month will have the same year but the next month may not.

The EPW file designates a start day of week for the weather file (ref: Data Period line). For flexibility, the RunPeriod field allows you to change that. And you can choose a week day for any subset run period you might select out of the weather file. Day of Week is primarily used for scheduled values, set points, etc.

As noted in the Input/Output Reference:

Field: Day Of Week For Start Day

For flexibility, the day of week indicated on the weather file can be overridden by this field's value. Valid days of the week (Sunday, Monday, Tuesday, Wednesday, Thursday, Friday, Saturday) must be entered in this field.

WARNINGS

During my simulation, I got this warning:

```
** Warning ** COIL:DX:COOLINGBYPASSFACTOREMPIRICAL "SOUTH_2_SYSTEM COOLING COIL" - Air-cooled  
condenser inlet dry-bulb temperature below 0 C. Outdoor dry-bulb temperature = -0.35
```

But there was no severe error. My two questions are:

1. Can I believe the results with this kind of warning?
2. What causes the program to issue such a warning and what does the warning mean?

Answer

Question 1:

You can verify if the warning is correct by looking at the Environment:Outdoor Dry Bulb and DX Coil Total Cooling Rate report variables. For the time that you see the warning, check that the DX coil is operating and verify that the outdoor dry-bulb temperature is above 0C. When the outdoor dry-bulb temperature is below 0C and the DX coil is operating, you will see this warning. If this warning is false, try downloading the new version of EnergyPlus 2.0, there was some work performed to make warnings more readable and reliable with 2.0.

Question 2:

EnergyPlus gives warnings to help the user perform simulations. Whether to help with setting up the input file with equipment node connections, or to warn the user that performance issues exist with the equipment selected for the simulation. This particular warning tells you that DX cooling equipment should not be operated at very cold outdoor dry-bulb temperatures. This type of operation can damage the equipment due to poor oil circulation. Although you cannot damage a DX compressor in a simulation, you can damage it in real life and this warning is meant to help you select the appropriate type of equipment for your application. Cooling is not typically required when the outdoor temperatures are very cold. If cooling is required when outdoor temperatures are very low, maybe a hydronic system would be better served in your climate.

ECONOMICS AND DEMAND COST CALCULATIONS

I need to figure out what portion of the utility bill is due to lighting, HVAC, process, etc. The billing is simple: demand and consumption charges. Since demand charges are by far the largest part of the bill, I wanted an easy way to split these charges among the various services. I'm not sure EnergyPlus can do this.

Answer

These calculations are not yet supported inside EnergyPlus; you would have to do them with postprocessing. However, a formal methodology for splitting demand charges across end uses was prepared as part of the commercial building tax credits. See section 3.1 of the report at:

<http://www.nrel.gov/docs/fy07osti/40228.pdf>

[Energy Savings Modeling and Inspection Guidelines for Commercial Building Federal Tax Deductions \(NREL/TP-550-40228 \)](http://www.nrel.gov/docs/fy07osti/40228.pdf)

M. Deru
National Renewable Energy Laboratory
1617 Cole Boulevard
Golden, CO 80401-3393

February 2007

Introduction

This document provides guidance for modeling and inspecting energy-efficient property in commercial buildings for certification of the energy and power cost savings related to § 179D of the Internal Revenue Code (IRC) enacted in § 1331 of the 2005 Energy Policy Act (EPAAct 2005) and noted in Internal Revenue Service (IRS) Notice 2006-52. Specifically, § 179D provides federal tax deductions for energy-efficient property related to a commercial building's envelope; interior lighting; heating, ventilating, and air conditioning (HVAC); and service hot water (SHW) systems.

Table of Contents

- 1 Introduction
- 2 Guidelines for the Interim Lighting Rule
- 3 Energy Modeling Guidelines
 - 3.1 Calculating the Energy and Power Cost Savings
 - 3.2 Software Requirements
 - 3.3 Weather Data
 - 3.4 Building Model Parameters
 - 3.5 Internal Loads and Schedules
 - 3.6 Use of Standard 90.1-2004 Appendix G
 - 3.7 Model Documentation
- 4 Inspection Guidelines
- 5 References
- Appendices
 - A – Lighting System Compliance Form
 - B – 2005 Calif Nonresidential ACM Approval Manual Infiltration Modeling Rules
 - C – 2005 Calif Nonresidential ACM Approval Manual Internal Loads and Schedules
 - D – 2005 Calif Nonresidential ACM Approval Manual Illuminance Categories and Luminaire Power

LATENT LOADS AND PURCHASED AIR

Does the purchased air simulation calculate the latent cooling loads?

Answer

Purchased Air is always supplied to the zone at the user-specified temperature and humidity condition. All moisture balances in the zone are included in the simulation, including the impact of infiltration and internal latent gains. However, Purchased Air does not actively control the zone humidity to a specific level.

Purchased Air Total Cooling Energy (or Rate) reports the latent and sensible energy required to bring the return air (plus outside air if specified) to the specified supply air temperature and humidity ratio.

Purchased Air Sensible Cooling Energy and Purchased Air Heating Energy report sensible energy only.

For definitions of Purchased Air report variables, please see the Input/Output Reference, p. 716 (pdf p. 755). For the equations used in Purchased Air, please see the Engineering Reference, p. 631 (pdf p. 667).

PURCHASED AIR HEATING AND COOLING

I'm trying to understand purchased cooling and heating. I simulated the model in PurchAirTables.idf and got Zone/Sys Sensible Heating Rate and Purchased Air Sensible Cooling Rate; however, there were different values for the same zone. Why?

Answer

These two values should be identical, and they are for most hours. However, after reviewing your file, there appears to be a slight difference in the first hour in the morning during recovery from setback. We will check on this.

Please bear in mind that the concept of Purchased Air was supposed to provide a simple way to calculate loads without simulating a full HVAC system. Purchased Chilled Water or Hot Water is commonly available from urban district heating and cooling systems. Purchased Air is analogous to this but delivered directly to the zone with no air handling system in the model. Purchased Air would not ever exist in the real world, but it is a convenient simulation concept.

PURCHASED COOLING VS FAN COIL COOLING

I'm getting results of Purchased Cooling being lower than the sum of the fan coil cooling rates.

Answer

"Fan Coil Total Cooling Rate" is the net cooling provided to the zone, including free cooling from outside air delivered by the fan coil unit. It is not the chilled water demand from the fan coil unit. For that, you would need to report "Total Water Cooling Coil Rate" or "Water Cooling Coil Chilled Water Consumption". Most report variables are defined in the Input/Output Reference. Search for the report variable name.

HOW TO CHANGE THE DEFAULT OUTPUT UNITS

In the report meter file, the unit of value in plant, facility, Building, etc., is kJ. I want to know the value-kw of system component electricity consumption. How do I get the value (kw unit)

Answer

The default output units for EnergyPlus energy meters is [J] (not kJ). For REPORT METER output, you can use the convertESOMTR utility to convert values to different units. See "convertESOMTR" in the Auxiliary Programs document. The default convert.txt file contains many active conversions to convert from SI to IP units. You can create your own convert.txt file which only converts electric meters to kWh.

EQUIPMENT SCHEDULING

I am modeling a building with an HVAC system that turns off during hours of non-occupancy. I have noticed, according to the ESO, that on most mornings the zones need to be cooled before occupancy. The HVAC system turns on one hour before occupancy and is very good at bringing the space to temperature before occupants arrive; however, it has a difficult time maintaining temperature during occupancy. I don't know the cause, unless I don't have the equipment scheduled during my design day sizing routines and this is affecting my cooling coil sizing.

Answer

As you suggest, internal gains, plus outside air, must be at expected peak levels during the cooling design day(s) in order to adequately size the coils. One quick check is to change the day type on your cooling design day(s) to be a Monday and see if this helps the situation. Also check that the supply air temperature controls are consistent with the temperatures in the zone and system sizing objects. And remember that fan heat is neglected in the current sizing algorithm. This can be allowed for by setting the SYSTEM SIZING cooling supply temperature a few degrees (equal to the supply fan temperature rise) lower than the ZONE SIZING cooling supply temperature.

COOLING SIMULATION

I'm simulating an 18-story building served by Desiccant CV. I've checked (and re-checked) the objects, fields, input values, nodes, node connections, etc., and everything seems to be okay. However, cooling isn't operating. The indoor radiant temperature is 40~50C. In the Report meter, the value of Cooling:electricity is 0 so cooling isn't operating.

Answer

The cooling is operating. See PurchasedCooling:Facility. Because the chilled water loop is served by PURCHASED:CHILLED WATER, there will be no Cooling:Electricity consumption. Some cooling is occurring but it is not enough. Report the chilled water outlet temperature. If this is not meeting setpoint, then the amount of purchased chilled water is too small. If the chilled water setpoint is being met, report the air temperature at the air loop outlet. If this is not meeting the setpoint, then the chilled water coil may not be large enough. If the air temperature is OK, then the airflows to the zones may not be large enough.

WORLD COORDINATE SYSTEM

I received this warning message:

```
** Warning ** World Coordinate System selected.  
** Any non-zero Building/Zone North Axes or non-zero Zone Origins  
** are ignored. These will be used in Daylighting:Detailed  
** calculations but not in normal geometry inputs."
```

In my simulation result, all daylighting reference points were outside the zone range to which they belonged. What does the word "ignored" mean in the context of the error message and how does it influence the simulation process?

Answer

Daylighting coordinates are always put in relative to zone origin coordinates. If you put in non-zero building/zone north axes or non-zero Zone Origins, then the Daylighting coordinates must be relative to them. If you put in your daylighting coordinates in world coordinates, they will likely be outside your zone boundaries and you will have problems.

If this is really the case, then you can set your north axis values (both Building and Zone) to 0 as well as the Zone Origin (X,Y,Z).

"Ignored" means that these won't be used when you input coordinates for your surfaces, windows, overhangs, etc. but WILL be used in calculating actual Daylighting reference points.

Question

So does your answer mean that I can get a correct daylighting simulation using only a relative system? Does that mean I will get a wrong building energy consumption (with daylighting) using the world coordinate system?

Answer

It is not correct to say that "a correct daylighting simulation result can be obtained only with relative system." If the building geometry has been set up using world coordinates, you can still use daylighting. The coordinates on the heat transfer surfaces honor the SurfaceGeometry rules for World or Relative coordinates. So, first, you must make sure that the surface coordinates are consistent with the geometry rule.

Daylighting reference points do not honor the setting in the SurfaceGeometry object. They are always relative to the zone origin. If the zone origin is 0,0,0, and the zone north axis is zero, then there is no difference between relative and world coordinates for that zone.

WARMEST FUNCTION MODELING

I am modeling the warmest function in a Hospital building. The model is equipped with a VAV system (pre-heat coil, cooling coil, a central heating coil, and reheat coils). I have the warmest setpoint node set at the supply fan outlet. The warmest function is set up to deliver supply air between 55F and 60F. Also, the chilled water is delivered to the cooling coils between 44F and 52F, depending on outside air temperature.

When I increase the upper limit of the warmest function from 60F to 65F and the chilled water upper limit from 52F to 54F, the natural gas consumption of the boiler increases by 7%. In addition, the supply fan energy usage increases by 6%. The energy saving benefit of the chiller and the cooling tower, 2% and 3%, respectively, is not enough to overcome the boiler and fan energy usage increases. The overall energy usage (total facility) increases by 2%.

I am confused to why this is happening. How can increasing the reset temperatures increase overall energy consumption instead of decreasing it? Why is the heating energy usage increasing in this manner?

Answer

The fan energy increase makes sense if you increase the supply air temperature (you cannot avoid this). And the chiller savings also make sense if you increase the chilled water temperature. A reduction in cooling tower energy would only make sense if the load on the tower was reduced. Since the chiller energy (and not the load in the space) is reduced, the load on the tower is reduced. For the natural gas consumption, check the minimum air flow fraction on the VAV reheat boxes. The natural gas consumption will only increase if the sensible cooling delivered to the space increases when the VAV boxes are at the minimum air flow rate. Check the zone sensible cooling rate for the zones that have increased VAV box reheat energy. Find a time when the zone VAV box is at the minimum air flow rate (you can also look at the VAV box damper position for this check) and compare this zone sensible cooling rate to what you get when you change the set points. You could also reduce the minimum air flow fraction on the VAV boxes and the natural gas consumption should decrease. This can also be accomplished by decreasing the maximum air flow rate on the VAV boxes that are using more reheat. You want this type of reheat to be at a minimum if at all possible.

The increase in gas use is not right. If there is a central heating coil in this VAV system, and it is picking up it's setpoint from the air loop outlet node, it may be warming the cool mixed air so that some VAV damper somewhere can stay fully open at low cooling loads. Be sure the heating coil setpoint is not linked in any way to what the warmest set point manager is doing for cooling. No two setpoint managers should be setting a temperature setpoint on the same node.

U-VALUE OF MATERIAL USED

How can I find out the U-value of all the materials used in a model ?

Answer

EnergyPlus documentation is searchable; if you enter U-Value, you will find numerous examples. The nominal U-value in each surface (pdf p. 30 in the OutputDetails and Examples document) using the "Report, Surface, Details;" command as well as the "Report, Constructions;" command gives the "nominal thermal

conductance" for each construction along with other information about the construction. You can also add a "Report, Materials;" command for similar information about each material referenced (whether used or not).

SYSTEM AVAILABILITY MANAGER

Is it possible to turn off only the chiller equipment part of the time and use outside air to maintain the temperature? I want to turn off the chiller and use outside air, via the supply fan, whenever the outside air reaches the appropriate temperature (free-cooling/mechanical ventilation). I'm using an electric chiller with an air-cooled condenser. When I tried using the system availability manager the entire system is turned off, including the fans, causing all sorts of problems.

Answer

You can specify a system availability manager for the chilled water plant loop. See PLANT LOOP "Chilled Water Loop Secondary Water Loop" in the example file IceStorage-Parallel.idf. However, if free cooling (outside air economizer) is enough to meet the cooling load, this should not be necessary. The cooling coil will not turn on and the chilled water loop will shut down (if the pump control is INTERMITTENT).

NATURALLY VENTILATED GREENHOUSE

I'm starting a project on a greenhouse that is naturally ventilated through openings at the top and at the bottom. The zone is 8 meters high. Do you think EnergyPlus can handle the thermal stratification and the buoyancy effect properly?

Answer

If you use the Airflownetwork model (based on AIRNET), the buoyancy effect for bulk ventilation flow can be modeled rather well, but stratification is not part of this model.

FREE-FLOATING SIMULATION

Can I simulate a free floating condition (no HVAC) with EnergyPlus?

Answer

The 1ZoneUncontrolled example file illustrates the free floating condition (no HVAC). Under purchased or other HVAC systems, you can use the control type "0" in your schedules during the times you want the system to float.

DIRECT AIR MODEL

To model an AirLoop with Direct Air Terminal; is there an example file?

Answer

The Direct Air Terminal is a constant volume air distribution unit. This can be used with either constant or cycling fan air systems. Furnaces, Heat Pumps, and Unitary Systems would use this type of air distribution unit. From the Engineering Reference on pg 392:

Direct Air

Central system air is usually supplied to a zone through a terminal unit such as a single duct VAV reheat box. Sometimes, however, it is desirable to supply central system air directly to a zone without any zone level control or tempering. An example would be Furnace or Central DX equipment. Direct air is the component used to pass supply air directly into a zone without any thermostatic control. The Direct Air unit allows the program to know what zone this branch of the air system is attached to and a place to input the maximum air flow rate.

The Direct Air object creates the capability of supplying central system air directly to a zone and only contains the zone inlet node. This node is both the zone inlet node and the outlet node of the Zone Splitter. It can be thought of as a balancing damper in the duct branch going to the zone. This inlet flow rate can be adjusted by a schedule. This can be thought of as a seasonal adjustment of the balancing damper.

For the Direct Air objects to work correctly, it is important in any systems including Direct Air objects for the sum of the maximum zone air flow rates to be equal to the maximum central system flow rate. The zone maximum flow rates are specified in the direct air inputs. The central air system flow rate is specified in the Air Primary Loop input and also in the air loop branch and central fan inputs.

OUTSIDE AIR AND DX COILS

I am having a hard time getting EnergyPlus DX coils to handle a 100% outdoor air flow. I am using a 100% outside air stream as the DOAS air stream, so I want the temperature to be neutral (18C), and I have a pre-defined volume airflow. I tried putting two multimode coils in series and two standard bypass factor coils in series. In both situations, I get a ton of warnings when I run the simulations, namely that the cfm/ton is out of range during the simulations. I must be setting something wrong. Is there a way to set up DX coils so that I can meet that 18C exit temperature without the warnings?

Answer

Assuming this is a constant volume system, two DX coils in series will work. The coils need to fall within the allowable flow/capacity range (0.00004027 to 0.00006041 m³/s per watt of rated total cooling capacity, 300 to 450 cfm/ton). For example, for a flow rate of 4500 cfm, each coil could have a capacity of 10 to 15 tons. If this is not a constant volume system, then you will need to use Coil:DX:MultiSpeed:CoolingEmpirical with the same constraints at full flow.

Question

There are two BypassFactorEmpirical coils in series in a constant volume system. I've hand sized them to be within the cfm/ton limits (as near as I can tell) and I am still getting warnings about being over those limits (high air flow).

Answer

The problem is not the dual DX coils. Even with just one DX coil, the errors happen. For some reason some early iterations at certain time steps are throwing a flow rate >8 m³/s at the ventilation system mixed air outlet. The reported flow rate for this node is always 3.781 as it's supposed to be. But when the DX coils see the higher flow rate, the warnings are tripped.

*.RVI AND *.CSV

I'm trying to force the contents of certain variables to a specific .csv column. This would allow me to process the values with the Excel file using very basic cell referencing. I have seen some discussions about using the .rvi file to force the column order of variables listed in .csv file.

Answer

You must use the .rvi file for this purpose. Whatever you put in report variables gets "shoved out" to the .eso file (or in case of meters also the .mtr file) as the information emerges in the simulation.

The actual order for that would be shown in the first part of the .eso file (before the part that says "End of Data Dictionary").

In the .rvi file, you can either specify the generic form of the report variable, e.g.:

```
Zone/Sys Air Temperature
Zone/Sys Sensible Heating Energy
Zone/Sys Sensible Cooling Energy
```

or be really specific:

```
Zone 1:Zone/Sys Air Temperature
Zone 1:Zone/Sys Sensible Heating Energy
Zone 1:Zone/Sys Sensible Cooling Energy
```

The first instance will give you all the zones, Zone/Sys Air Temperature, followed by all the Zones, Zone/Sys Sensible Heating Energy, etc. The second instance will put the three variables by zone. There are also ways of extracting the hourly or monthly values in a .csv file even though the .eso file may contain multiple hourly and

monthly mixed. You might want to explore using the utility XESOVView (not part of the EnergyPlus install). Information on XESOVView is on the Yahoo groups website and at www.energyplus.gov.

SIMULATING DOUBLE SKIN FAÇADE

I'm simulating an office with a double skin façade using the "exterior vented natural cavity."
The façade is naturally ventilated and has Venetian blinds in the gap. The blinds are exposed to solar radiation and, during the day, they can reach very high temperatures which may affect the temperature of the air in the gap and/or the performance of the glass façade. Does EnergyPlus take into account these two effects?

Answer

The exterior vented natural cavity model is for *opaque* surfaces only and does not apply to glazing systems. There is a model for blinds in the window gap that takes those two things into account, but I am not sure if it works together with the air flow window model.

Question

Thanks for your answer. I compared two simulations, one with the façade as exterior vented cavity and the other with the façade as a simple window, and it seemed that solar transmission through the glazing system is the same in both the cases. Even if the exterior vented cavity is only for opaque surfaces do you think it is possible to use the model for a glass façade, using as input the data of the glazing system (thermal emissivity and solar absorptivity)? Which differences or errors would occur using that model?

Answer

If I understand you correctly, I think what you are actually getting in the model is a regular window surrounded by a wall with a double skin. See if the inside face wall surface temperatures are different. The window shouldn't be changing.

The exterior vented cavity model, which assumes that all the solar is blocked by a thin, solid layer on the outside, cannot be used for glazing. Opaque means no transmission. Inside EnergyPlus this becomes a special boundary condition for surface heat transfer modeling that is not available for window modeling. The window still gets exterior environment boundary conditions even though its parent surface has the exterior cavity boundary condition.

Double façade improvements are being planned for the future.

Possible Workaround

if you really wanted to use the exterior vented cavity model on your DF, it could apply to a case where the double façade has totally opaque shades that are closed tight. You would use regular materials in the construction for the surface that approximate the DF (no subsurface and no glazing materials). The baffle in the exterior cavity model can get quite hot and then re-radiate infrared (but no shortwave) and convect to the underlying surface. But note that air gap models are much better developed for windows than for regular constructions; you would have to come up with thermal properties for each layer. One advantage of this approach might be a better model of transient behavior because the window models are always steady state.

See also: EnergyPlus Technical Note – Opaque Ventilated Façades (July 2007 User News)

On May 28, 2007, Emanuele Naboni (PhD Candidate), a visiting researcher from the Politecnico di Milano (Italy) gave a lecture at Lawrence Berkeley National Laboratory on how he used EnergyPlus to model Ventilated Opaque Façades. A .pdf of his presentation is available at

<http://SimulationResearch.lbl.gov/reports.html> under the heading "All EnergyPlus Reports."

Or, here is the direct link: http://simulationresearch.lbl.gov/dirpubs/vent_facade.pdf

WINDOW5 DATA VS IDF

I use a Window5 data file for my windows properties, and I would like to know what the best strategy is to model the window (detail vs accuracy vs time).

In the IDF file, I modeled the windows as a long, continuous, horizontal strip in the external wall. However in Windows5 I modeled the windows in the correct dimensions (glass+frame).

I initially thought that the Window5 data would take precedence over the IDF dimension. But referring to the manual, I found that this is not true. When there is a discrepancy in the dimension between IDF and Window5, then IDF dimension will take precedence. The glass properties will be taken from Window5, but the dimension will be from IDF file. But what about the frame? The manual does not mention anything about the frame, will the properties be taken from the Window5 file or from IDF?

This brings the question back to the best way to model the window. If we want to justify, for example, that a certain window is so good that we do not need the perimeter heating, then we need to model the frame as detailed as possible. Do we really need to model every window exactly to its dimension? This will increase the simulation run time tremendously.

Answer

You are correct that the dimensions of your window, as input in the IDF, will take precedence over the dimensions of the window in the Window5 data file, *UNLESS* the window in the Window5 data file contains two sections that have different glazing systems, in which case the Window5 dimensions will be used. See details of this under "Importing Windows from Window 5" in the "Group - Thermal Zone Description/Geometry" section of the Input/Output Reference.

The frame that is part of the Window 5 data file will be used to create a WindowFrameAndDivider object that will be assigned to the window for EnergyPlus analysis. EnergyPlus will generate a warning if the dimensions of the window in your IDF differ from the Window5 dimensions by more than 10%. This warning indicates that some inaccuracy may result because of the different area and perimeter relationships between the glass area and the frame. Unless you are especially concerned with the frame effects, you should create a simple narrow frame in your Window5 file to reduce these differences. It is not possible to create a Window5 window system for export to EnergyPlus without including some type of frame.

The importance of modeling your windows precisely depends on what you are trying to analyze. If you are interested in daylighting effects, then it is important to model the windows accurately. If you are only looking for general thermal effects, then this is not as important.

AIRFLOW NETWORK FAN

I am starting to use the AirFlow Network with only the MultiZone:Surfaces, referring to the SIMPLE house example. From the .err file, a zone needs two surfaces and I have added a door to each zone. So, correct me if I am wrong, but my understanding is that the air in the AirFlow Network goes through ceiling crack/leakage and door leakage. I did not use Direct air or fan as in the Simple House example but I do have a Window AirConditioner in the second zone.

The error message included this statement:

```
GetFanIndex: Fan not found=Zone2WindACFan
GetAirflowNetworkInput: The fan type defined in
AIRFLOWNETWORK:DISTRIBUTION:COMPONENT CONSTANT VOLUME FAN =
ZONE2WINDACFAN is not allowed etc.
```

I assume that the 'Fan' in the error message refers to the fan in Window AC. But how do I enter the index of this fan or overcome the errors in the .err file?

Answer

After reviewing your input file, it doesn't look like you specified a fan as Zone2WindACFan; that is what generated the first error message (GetFanIndex: Fan not found=Zone2WindACFan). In other words, Zone2WindACFan is not listed in fan index. You may define the fan in the AIRFLOWNETWORK:DISTRIBUTION:COMPONENT CONSTANT VOLUME FAN object. Since the fan is not listed in the fan index, the program could not continue.

My suggestion is to disable the AirflowNetwork model first to see whether you can eliminate the first error. The model disablement is to use the choice of NO MULTIZONE OR DISTRIBUTION in the AirflowNetwork Control field in the AirflowNetwork Simulation object.

By the way, Window AC is not supported for the time being if you use the choice as MULTIZONE WITH DISTRIBUTION or MULTIZONE WITH DISTRIBUTION ONLY DURING FAN OPERATION in the AirflowNetwork Control field in the AirflowNetwork Simulation object

Question

Many thanks for the information on how to disable the AirFlow Network.

I will try Infiltration first. Do you have any advice on which variables to use for this object if the infiltration to the attic (space between the horizontal overhang ceilings and the sloping roof) is only through the corrugations in the roofing metal sheets as they are nailed to a horizontal wood?

Meanwhile, is Compact HVAC:System:Unitary supported if I use the choice as MULTIZONE WITH DISTRIBUTION or MULTIZONE WITH DISTRIBUTION ONLY DURING FAN OPERATION in the AirflowNetwork Control field in the AirflowNetwork Simulation object?

Answer

I don't have suggestions on attic infiltration. You may try to find values in other publications.

The AirflowNetwork model requires an air primary loop. Any coils and fan have to be a part of the loop. The allowed coil types are listed in the EnergyPlus .idd and Input/Output Reference. An example input file of AirflowNetwork_Multizone_House.idf uses UNITARYSYSTEM:HEATPUMP:AIRTOAIR. The unitary system uses allowed coil types. The answer to your question is that if a compact unitary system uses allowed coil types, it should work.

OVERLAPPING SHADOWS ERROR

When I run a yearly simulation for my office building, I get the following warning:

```
***** ===== Recurring Surface Error Summary =====  
** The following surface error messages occurred.  
** Too many figures in shadow overlap errors occurring.  
** These occur throughout the year and may occur several times  
** for the same surfaces. You may be able to reduce them by  
** adding DIAGNOSTICSDoNotMirrorDetachedShading;  
** Warning ** Too many figures in a shadow overlap  
** Overlapping figure=003>006>O1>WF0, Surface Class=[Wall]  
** This error occurred 10 times.  
** Figure being Overlapped=009-03E, Surface Class=[Wall]  
** The too many figures errors occurred 10 times (total)  
** EnergyPlus Completed Successfully-- 10 Warning; 0 Severe Errors;  
** Elapsed Time=00hr 03min 33.00sec
```

I don't really understand what this warning means and how it affects the results of the simulation. These two walls are both aligned vertically (ideally they can be on the same vertical plane), but 009-03E is an exterior wall on the 4th floor and 003>006>O1>WF0 is an interior surface located on the 2nd floor. I tried to erase all my shading elements, but I still got the same warning.

Answer

This error means that the shadowing calculations have overrun some predefined limits on the number of overlapping shadows on a single surface. In the SHADOWING CALCULATIONS object, there is a second field "Max_figures_shadow" which allows you to increase this limit. The default is 15000. Try a large value, such as 20000, and see if this solves the problem.

VENTILATION OBJECTS

I would like to have three ventilation systems serving the same zone; each one with a different control. However, I think that EnergyPlus allows a maximum of two ventilation objects to serve one thermal zone. Is this true? Can I get around the two object limit?

Answer

Actually, only one simple Ventilation object per zone is allowed. Multiple ventilation objects are on the wish list.

Here are some possible work-arounds:

Zone equipment (like Unit Ventilator) can serve as a second source of ventilation.

A full air loop system can serve as a third source of ventilation.

The simple Infiltration object can be considered the same as ventilation.

If you really need to, it is possible to do a modeling trick where an additional Ventilation object is used on a small dummy zone. The dummy zone is connected to the main zone using a pair of Mixing objects.

CURVED BLINDS

Is there a way to model curved blinds with EnergyPlus? Page 98 of the Input/Output Reference manual (4/9/07) states "Slat curvature, if present, is ignored."

Answer

It would not be exact, but you could use the "slat thickness" to model the curvature of the blind slat.

In this case the slat thickness could be found by laying the slat on a flat surface and measuring the distance from the slat midpoint to the flat surface. Refer to the Input/Output Reference document for a complete discussion of Material:WindowBlind.

EXTERIOR FIXED LOUVERS

I need to model a shading device with the following composition:

Approximately 20 fixed louvers that span the width of the window.

Louvers are arranged on a slope, where the uppermost louvre at the top of shading device is closer to the window face (237mm from the window face at a height level with the window head) than the louver at the base of the sunshade (986mm from the window face at a height of 529mm above the window sill).

Louvers between the uppermost and lowermost louvres are spaced evenly and linearly.

Should I simply model in the geometry of the shades directly and use Surface:Shading:Attached, or will that result in a ridiculous amount of processing time? Are there any ways to approximate the effect of a shading device like this that won't excessively compromise the accuracy of the simulation?

Answer

Modeling as Surface:Shading:Attached is certainly the most accurate approach. If this occurs on only one window, the computing time should not be excessive.

If the same configuration occurs many times around the building, then it might be a problem. When using this approach, the default action of EnergyPlus is to mirror every shading surface, so that you do not have to worry about the facing direction of the shading surfaces. This is necessary because surfaces in EnergyPlus only cast shadows in one hemisphere (in the direction of the outward facing normal).

If computing time becomes an issue, you can use the diagnostics command to turn off this mirroring: DIAGNOSTICS,DoNotMirrorDetachedShading. However, if you do this, you **MUST** make sure that the azimuth and tilt of the slats is facing down and toward the window. Also, if the slats are light colored, you should turn on reflections using the solar distribution field in the BUILDING object. And you can specify the reflectance properties using Shading Surface Reflectance objects.

An approximate alternative would be to treat this as an exterior blind with large slats. See MATERIAL:WINDOWBLIND and WindowShadingControl.

INDOOR AIR QUALITY

Where do I find information about CO₂, CO density?

Answer

For any information you want out of an EnergyPlus simulation, you look at the RDD (report Variable) and MDD (Meter variable) files. Also, check the "pollution calculations" in the Input/Output Reference.

As of V2.0 release, the simulations can also report:

HVAC,Sum,Carbon Equivalent Pollution From NO_x[kg]

HVAC,Sum,Carbon Equivalent Pollution From CH₄[kg]

HVAC,Sum,Carbon Equivalent Pollution From CO₂[kg]

Zone,Meter,Carbon Equivalent:Facility [kg]

Zone,Meter,CarbonEquivalentEmissions:Carbon Equivalent [kg]

Question

In the report field, I found the electric emission CO₂, CO, and other contaminants in report meter. What do these contaminants mean and is this also IAQ information?

Answer

No, those are emissions associated with the use of energy. They are not related to indoor air quality at the building site but refer to pollution generated back at the power plant. EnergyPlus can model the energy performance implications of some measures intended to address IAQ, but it does not directly model IAQ. You may want to try the software from NIST called CONTAMW. Here's a direct link to the CONTAMW page: <http://www.bfrl.nist.gov/IAQanalysis/CONTAM/index.htm>. For further reference, here is a list of all downloadable software from the Building and Fire Research Laboratory at NIST at <http://www.bfrl.nist.gov/info/software.html>.

REPORTING CONTAMINANTS

Can EnergyPlus give me a detailed report of contaminants released from a building?

Answer

Please see "Report Environmental Impact Factors" in the Input/Output Reference (starting on p. 1351, pdf p. 1390) and the Engineering Reference (starting on p. 817, pdf p. 853). These report variables and meters simply take the environmental factors input by the user for electricity and fuels and multiply them by the site consumption of each particular energy source. Equations are currently not in the documentation, but they are all similar to the following.

For example, a building purchases electricity and natural gas, and has no on-site power generation

$$\text{CO2:Facility} = (\text{CO2EmissionFactorForElectricity} * \text{Electricity:Facility}) + (\text{CO2EmissionFactorForGas} * \text{Gas:Facility})$$

where

Electricity:Facility and Gas:Facility = the consumption meters for electricity and natural gas

CO2EmissionFactorForElectricity and CO2EmissionFactorForGas = the "CO2 emission factor" field in the Fuel Factors object. There are datasets of fuel factors in the Datasets folder: Electricity USA Environmental Impact Factors.idf and Fossil Fuel Environmental Impact Factors.idf .

PURCHASED AIR AND INFILTRATION

I'm simulating a module with only one external, insulated wall that contains a double glazed window, which takes up 10% of the floor area. The rest of the walls, floor and ceiling are adiabatic. The orientation is South and there are no equipment or light loads. The model includes a constant infiltration flow rate of 0.04 m³/s, and uses Purchased Air without limits. Thermostat settings are 23.5 for cooling and 19.5 for heating (using dual setpoint with deadband). The system can operate 24 hours. Certain summer night hours indicate an outside temperature of 20C. However, for those same hours, and despite the infiltration rate, I get a report that the Purchased Air operated cooling in order to reach the setpoint. Is that possible?

Answer

It is quite possible, depending on how much heat gain your cell receives during the day and storage in the building elements. You could remove (turn off) the purchased air all together and allow the cell to float -- to see how high the temperature gets without benefit of cooling.

DOE-2 TRANSLATOR FOR ENERGYPLUS

To translate DOE-2 models to EnergyPlus, download these zip files from the Yahoo group files area.

http://groups.yahoo.com/group/EnergyPlus_Support/files/Release_pieces/DOE2Translator.zip

Caveat -- this translator only works for DOE-2.1E input files and is limited to building envelope, internal gains, schedules, etc. It does not translate HVAC systems.

SOLAR WATER HEATING SYSTEM

I'm trying to find an idf example file of a hydronic system fed by a solar water heating system.

Answer

Example file SolarCollectorFlatPlateWater.idf serves a hot water storage tank with solar collectors. In this example the "use" side of the water heater feeds domestic hot water objects, but it could also feed hydronic coils by adding them to the demand side of the PLANT LOOP currently named "DHW" in the example.

SHGC AND SOLAR TRANSMITTANCE

I'm simulating a zone with a glass façade and I want to have 0.15 for the SHGC of the window:
I have run two simulations:

1st case: window with SHGC=0,15 solarTransmittanceNormIncidence=0,06

2nd case: window with SHGC=0,15 solarTransmittanceNormIncidence=0,10

Basically the amount of solar heat entering in the zone is the same (since the SHGC is the same), but the amount of solar radiation (beam + diffuse) is different. Does Energy Plus make any difference for calculating the cooling rate? And if I want to estimate the cooling peak value, which case should I consider?

Answer

EnergyPlus will calculate things differently for the two windows.

Keep in mind that SHGC is defined under a set of standard conditions. Two different windows could line up to have the same SHGC under the standard conditions, but won't necessarily perform equally at all times during annual simulation.

In general, there will be more time lag associated with the radiated portion of solar heat load because it will arrive first at inside surfaces of the zone. The portion of heat that is convecting off the inside face of window will go straight into the zone air and show up as load right away.

You would have to run both windows to see which has the higher peak. My guess is the first case would have higher peak, but it really depends on the particulars of the zone in question.

CHILLER OPERATION

In plant equipment, the simulation model contains two chillers, one large and one smaller. Generally, the larger chiller operates from morning to noon and the smaller chiller operates from afternoon to night. When calculating the energy consumption to the chiller electricity, should I sum those two values?

Answer

Each chiller will have an output report variable for its "Chiller Electric Power [W]" and its "Chiller Electric Consumption [J]" (see Input/Output Reference, section "Generic Chiller Outputs" and section "Report Variable"). Yes, you should sum the values for each chiller if you want to know the total electric power or electric consumption for the two chillers. As an alternative, the "Chiller Electric Consumption [J]" is metered (i.e., summed for all electric chillers) on Electricity:Plant. However, this meter will also contain the pump energy and cooling tower fan energy if a tower is being simulated. Look in the Input/Output Reference at the section entitled "Report Meter, Report MeterFile Only" for more information regarding report meters.

DX COOLING PROBLEM

We have an input file that uses the COIL:DX:CoolingBypassFactorEmpirical object and is causing some problems. When run through EnergyPlus (V. 1.4) an error is reported as follows:

```
** Severe ** COIL:DX:COOLINGBYPASSFACTOREMPIRICAL "WC:C2 FF OPEN" -  
** coil bypass factor calculation did not converge after max iterations.  
** The Rated SHR of 0.571, entered by the user or autosized (see *.eio  
** file), may be causing this. The line defined by the coil rated inlet  
** air conditions (26.7C drybulb and 19.4C wetbulb) and the Rated SHR  
** (i.e., slope of the line) must intersect the saturation curve of the  
** psychrometric chart. If the RatedSHR is too low, then this  
** intersection may not occur and the coil bypass factor calculation will
```

**** not converge. If autosizing the SHR, recheck the design supply air
** humidity ratio and design supply air temperature values in the System
** Sizing and Zone Sizing objects. In general, the temperatures and
** humidity ratios specified in these two objects should be the same for
** each system and the zones that it serves.**

Given the absence of a system sizing object, it is unclear how to address this problem.

Answer

The supply air condition in the zone sizing object is too low for the coil's process line to cross the saturation curve. You are using a rated inlet air dry-bulb temperature of 26.7 C and rated inlet air wet-bulb temperature of 19.4C (rated inlet air humidity ratio = 0.01125 kg/kg) with a zone sizing object supply air temperature of 16C and zone sizing supply air humidity ratio of 0.008. If you draw a line from the rated inlet air condition to the zone sizing supply air condition, you will see that this line does not cross the saturation curve on the psychrometric chart. Try increasing the supply air humidity ratio to 0.009. This should allow the bypass factor calculation to converge. Please upgrade to the current version.

RUN TIME FRACTION

We want to use Energy Plus to look at a typical heating and cooling system runtime at different load conditions. The model consists of a single story residence with a constant volume residential forced air furnace (FURNACE: HEATCOOL), a DX cooling coil (COIL:DX:CoolingBypassFactorEmpirical), and an electric resistance heating coil (COIL:ELECTRIC:HEATING). We are modeling the system using a cycling fan schedule.

To estimate compressor runtime over the (one-hour) simulation time step, should we use the cooling coil runtime fraction? We have run the simulations using different internal and ambient load profiles and the runtime fractions vary from 0.02, 0.06 to 0.1. There is some confusion about what the runtime fraction actually represents. Does it represent the fraction of the time step that cooling is on, or is it the rate at which the compressor cycles?

Answer

The runtime fraction (RTF) is the fraction of time the compressor operates during a time step. Upon review of the runtime fraction examples you gave, 0.1 is only 10% of the runtime fraction or approximately 10% of the DX coil capacity. This unit is over-sized for the load; make your coil smaller so the RTF is higher and near 0.9 at the design load.

SURFACE OUTPUT VARIABLES

The Opaque surface Inside Face Conduction[w] variable can be reported, but how do I get the Outside Face Conduction[w] variable value?

Answer

The documentation is searchable from the main menu. As it says in the Input Output/Reference: *Output variables applicable to opaque heat transfer surfaces (FLOOR, WALL, ROOF, DOOR).*

Note – these are advanced variables – you must read and understand the descriptions before use – then you must use the Diagnostics object to allow reporting.

Zone,Average,Opaque Surface Inside Face Conduction[W]

Zone,Average,Opaque Surface Inside Face Conduction Gain[W]

Zone,Average,Opaque Surface Inside Face Conduction Loss[W]

Zone,Average,Zone Opaque Surface Inside Face Conduction Gain[W]

Zone,Average,Zone Opaque Surface Inside Face Conduction Loss[W]

Zone,Average,Opaque Surface Inside Face Beam Solar Absorbed[W]

And under the Diagnostics object, see *DisplayAdvancedReportVariables*.

Use this to be able to use the “Opaque Surface Inside Face Conduction”; the name may be misleading to some. If you put in this field, then you will be able to report on this variable.

So, you need to add Diagnostics, DisplayAdvancedReportVariables; to get this variable reported.

ZONE SIZING OUTPUT FILE

When running a design-day-only simulation for the purpose of checking zone sizing, the ****cZsz.csv file has the last two lines labelled "Peak" and "Peak Vol Flow." For the mass flow rate columns, these values differ, and the "Peak Vol Flow" value is always less than the "Peak." What is the definition for these two values (I have NOT assigned a value for the "heating max air flow")?

Answer

The "peak" value is the maximum of the individual timestep mass flow rates. It is in kg/s, as are the timestep numbers. The "Peak Vol Flow" is the peak converted to volumetric flow rate: its units are m³/s.

Question

And at what temperature and pressure is the volumetric flow rate calculated? I get slightly different values using STP conditions. How about documenting this and modifying the .csv file to add the units in the description?

Answer

We'll modify the Zsz.csv file to include the units for "peak vol flow." The air density is for dry air at 20C, pressure = 101.325*(1-2.25577E-05*Elevation)**5.2559; this is an elevation corrected standard pressure formula from the ASHRAE Handbook of Fundamentals.

HOT WATER RADIATORS

I'm trying to connect hot water radiators using Energy Plus and I am getting some error messages I don't understand. I tried these connections (having already filled in the fields of CONTROLLED ZONE EQUIPEMENT CONFIGUGATION and ZONE EQUIPMENT LIST):

```
BASEBOARD HEATER:Water:Convective
Baseboard Name      HouseBaseboard
Available Schedule  House_Zone_OPERATION
Inlet_Node          HouseReheatWaterInletNode
Outlet_Node         HouseReheatWaterOutletNode
UA                  500
Max Water Flow Rate 0.0013
Convergence Tolerance 0.001
```

And I am getting the following error messages:

```
** Severe ** Node Connection Error, Node="HOUSEREHEATWATERINLETNODE",
Inlet node did not find an appropriate matching "outlet" node.
** ~~~ ** If this is an outside air inlet node, it must be listed in an OUTSIDE AIR
NODE or OUTSIDE AIR INLET NODE LIST object.
** ~~~ ** Reference Object=BASEBOARD HEATER:WATER:CONVECTIVE, Name=HOUSEBASEBOARD
** Warning ** Node Connection Error for object BASEBOARD HEATER:WATER:CONVECTIVE,
name=HOUSEBASEBOARD
** ~~~ ** Hot Water Nodes not on any Branch or Parent Object
** ~~~ ** Inlet Node : HOUSEREHEATWATERINLETNODE
** ~~~ ** Outlet Node: HOUSEREHEATWATEROUTLETNODE
```

I don't understand why "inlet" node did not find an appropriate matching "outlet" node. How do I correct the connection? Is there an example file??

Answer

What you have so far is fine, but it is not enough. There must be a complete PLANT LOOP and more components, including a pump and boiler, in order to serve the radiator. Please see example file WindACAuto.idf. This example has three radiators on the demand side of the plant loop, but you can construct your input to be similar with a single radiator.

Question

Thanks for your reply. To clarify, is a pump, a boiler, a radiator, a radiator thermostat and a room thermostat enough for the simplest possible modeling of a hot water radiator system?

Answer

Yes, if the pump is variable speed. If the pump is constant speed, then you need a splitter, a bypass pipe, and a mixer on the demand side. And by "radiator thermostat", I assume you mean a SET POINT MANAGER:* object to control the hot water supply temperature.

Question

I noticed that in the example file you suggested, it uses Purchased:Hot Water. Is this instead of using a boiler?

Answer

Yes, the example file uses purchased hot water (district heating) instead of a boiler. You can substitute a BOILER:SIMPLE where you see PURCHASE:HOT WATER referenced.

INDEX

Air		DOAS	27
Airflow Network Model	25	Doe-2 To Energyplus Translator	58
Constant Deck	17	Domestic Hot Water	7
Direct Air Model	51	Door-In-Wall	22
Fresh Air, Fan Coil System	9	Dx	
Humidity Control	25	Coil, Outside Air	52
Latent Loads And Purchased Air	48	Cooling Coil	22, 59
Mundt Model	27	Cooling Coil, Warnings	46
Outside	26	Dxf Drawing	26
Outside Air And Dx Coils	52	Economics, Demand Cost Calculations	47
Purchased	26	Envelope, Light	38
Purchased Air, Heat, Cool	48	Epw Files	33
Purchased Air, Ventilation	17	Equipment Schedules	49
Purchased, Infiltration	58	Error	
Reporting Iaq	57	Overlapping Shadows	56
Supply/Return Fan	11	View Factors	37
Window Air Conditioner	27	Facades	13
Air Partitions In Zone	27	Double Skin	53
Airflow Network		Fan	
Fan	55	Coil W/Fresh Air	25
Model	25	Supply Fan Variables	41
Airflow Object	35	Supply/Return Flow Rate	11
Autosizing		Floor Geometry	43
Chillers	36	Floor/Roof Tilt	15, 16
Errors	24	Flow Rate, Zero Air Loop Mass	45
Model Creation	35	Free-Floating Condition	51
Parametrics	36	Geometry	
Blinds		Dxf Drawing	26
Interior	34	Floor	43
Curved	56	Glare Index	21
Boiler, Residential	8, 11	Greenhouse, Natural Ventilation	51
Branch, Loop	5	Ground Temperatures	40
CFM/TON Limits	45	Heat Pump	10, 14, 43
Chilled Beam System	16	Heat Transfer	40
Chiller, Energy Consumed	59	Heater Element	44
Coefficient		Heating	
Convection	38	Solar Water System	58
Inside Convection	40	Hot Water Reset	39
Otherside	12	Hours Not Met	29
Wind Pressure	38	Humidity Control	25
Connected Surface	6	Idf Data Vs Window5	54
Contaminants	58	Illuminance Map	16
Contiguous Zones	14	Indirect Evap Cooler	43
Controls	5	Infiltration And Purchased Air	58
Convection Coefficient	38	Inside Convection Coefficient	40
Cooling		Interior Blinds	34
Purchased Vs Fan Coil	48	Light Shelf	18
Simulation	49	Loads	
Tower	12	Latent Loads And Purchased Air	48
COOLING		Loads Not Met	29
Cooling Coil Error	44	People, Lights, Etc.	18
Corridor Modeling	33	Reporting	37
Csv		Loop	
How To Specify	52	Branch	5
Zone Sizing Output	61	Loop, Zero Air	45
Cycling Time Step	43	Louvers	57
Daylighting	6	Macros	8
Obstructions	9	Material	
World Coordinate System	49	Phase Change	20
Design Volume Flow Rate	35	U-Value	50
Direct Air Model	51		

Meters		Floor/Roof.....	15
Change Default Units.....	48	Translator, DOE-2 To Energyplus.....	58
Custom.....	18	Turbulence, Induced.....	40
Reports.....	19	Unentered Other Zone.....	33
Natural Ventilation, Greenhouse.....	51	Unit Conversion.....	28
No Hvac, How To Simulate.....	51	Unitary Equipment.....	5
Otherside Coefficient.....	12	U-Value Of Material.....	50
Output, Change Units.....	48	Ventilation Objects.....	56
Parametric.....	8	View Factors Error.....	37
Parametrics		Warmest Function Modeling.....	50
Autosizing.....	36	Warning	
Phase Change Material.....	20	Connected Surface.....	6
Precipitation Data.....	21	Dx Cooling Coil.....	46
Purchased Air		Hot Water Radiators.....	61
Heat/Cool.....	48	Water-Side Economizer.....	17
Infiltration.....	58	Weather	
Rdd Special Model.....	43	Doe-2 To EPW.....	11,34
Reports		EPW Files.....	33
Contaminants.....	58	Modify.....	20
Electric Energy Demand.....	19	Precipitation Data.....	21
Iaq.....	57	Sky Temperature.....	22
Loads.....	37	WET BULB.....	45
Meters.....	19	Wind Pressure Coefficients.....	38
Outside Face Conduction.....	60	Wind Speed.....	36
Residential Heater Sizing.....	43	Window	
Room Air Node Temp.....	27	Air Conditioner.....	27
Run Period, Start Day.....	45	Conductivity.....	20
Run Time Fraction.....	60	Curved Blinds.....	56
Rvi		Exterior Fixed Louvers.....	57
How To Specify.....	52	Facades.....	13
Schedules.....	8, 10	Fatal Error.....	6
Equipment.....	49	Glass Variables Defined.....	12
Sensible Heating, Cooling.....	40	Interior Blinds.....	34
Set Point Temperature.....	24	Outside Reveal Shadowing.....	31
Shading Devices.....	29	Reveals.....	31
Shadows, Overlapping.....	56	Shading.....	6
Sky Temperature.....	22	Shading Device.....	21
Slab Program.....	40	Shading Devices.....	29
Solar Output Variables.....	39	Shgc, Solar Transmittance.....	59
Solar Transmittance.....	59	Switchable Glazing.....	34
Solar Water Heating System.....	58	Window Ac Vs Unitary.....	24
Start Day.....	45	Window5 Vs Idf.....	54
Supply Fan Variables.....	41	World Coordinate System.....	49
Switchable Glazing.....	34	Zone	
System		Air Partitions.....	27
Availability Manager.....	5	Contiguous.....	14
Direct Outside Air.....	27	Controls.....	41
Ventilation Objects.....	56	Corridor.....	33
Volumes.....	37	Heat Balance Equation.....	7
System Availability Manager.....	51	Multiplier.....	14
Tall Buildings.....	10	No Walls.....	32
Temperature		Partitions.....	26
Set Point.....	24	Sizing Output File.....	61
Therman Comfort.....	40	Unentered Other Zone.....	33
Thermostatic.....	40	Uniform Zone Multiplier.....	15
Thermostats.....	41	Volume.....	26
Tilt			